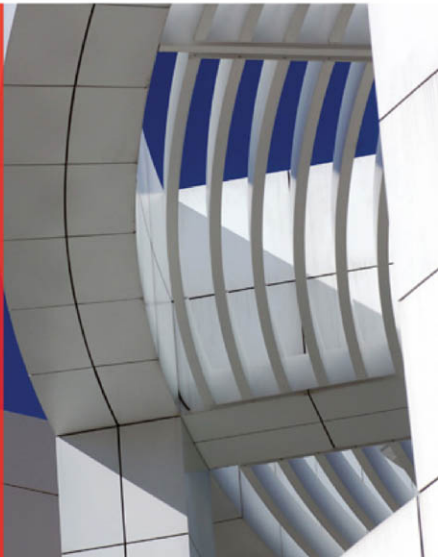


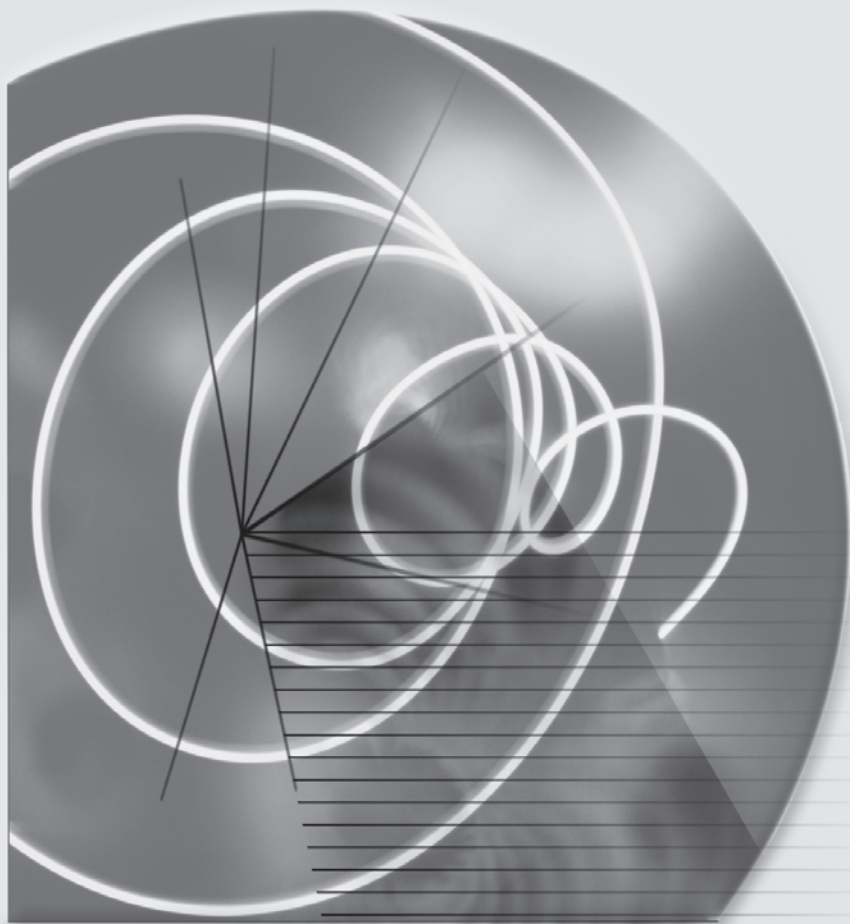
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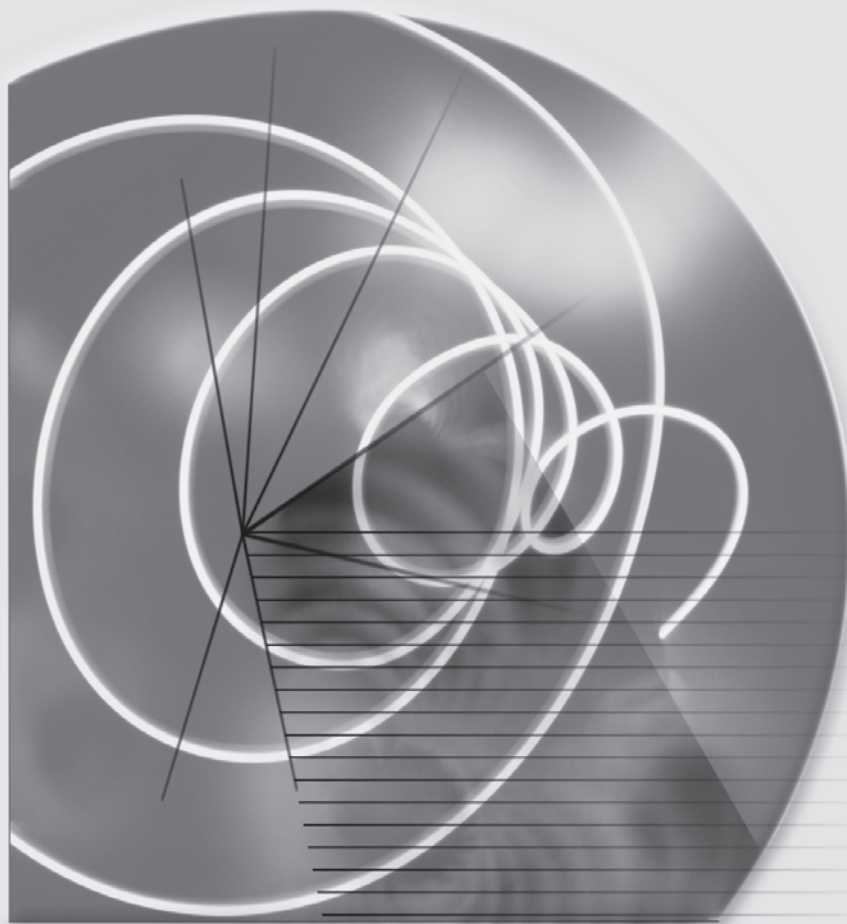
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With more than four million users around the world, AutoCAD offers engineers, architects, drafters, interior designers, and many others, a fast, accurate, and versatile drafting and modeling tool.

Now in its 17th edition, *Using AutoCAD 2010* makes using AutoCAD a snap, by presenting easy-to-master, step-by-step tutorials covering AutoCAD's commands.

BENEFITS OF COMPUTER-AIDED DRAFTING

AutoCAD is efficient and versatile for creating drawings. Some of its advantages include the following:

Accuracy — AutoCAD drawings are created and plotted to an accuracy of up to fourteen digits.

Speed — AutoCAD operators can easily copy and array objects, and otherwise edit the drawing. When operators customize the system for specific tasks, the speed of work increases even more markedly.

Consistency — AutoCAD is consistent in its methodology, and so the problem of individual style is eliminated. Companies can have a number of drafters working on the same project and produce a consistent set of drawings, when they follow established CAD standards.

Popularity — AutoCAD is the most popular software for creating and editing drawings. After learning to use AutoCAD, students are more likely to be hired; firms that use AutoCAD are more likely to find suitable new hires.

CAD APPLICATIONS

AutoCAD is being applied to many industries. The flexibility of this software has a major impact on how tasks are performed in architecture, engineering, interior design, manufacturing, mapping, piping design, and entertainment. A brief discussion of each of these applications follows.

Architecture

CAD allows architects to formulate designs in a shorter period of time than by traditional techniques. The work is neater and more uniform. The designer can use 3D modeling to help the client better visualize the finished design. Changes can be made and resubmitted in a very short time. Architects can assemble construction drawings using stored details.

AutoCAD's database allows architects to extract information from the drawings, perform cost estimates, and prepare bills of materials.

Many third-party applications help architects customize AutoCAD for their discipline. Software is available for doing quick 3D conceptual designs, providing building details, generating automatic stairs, and much more. www.autodesk.com/autocad

Revit

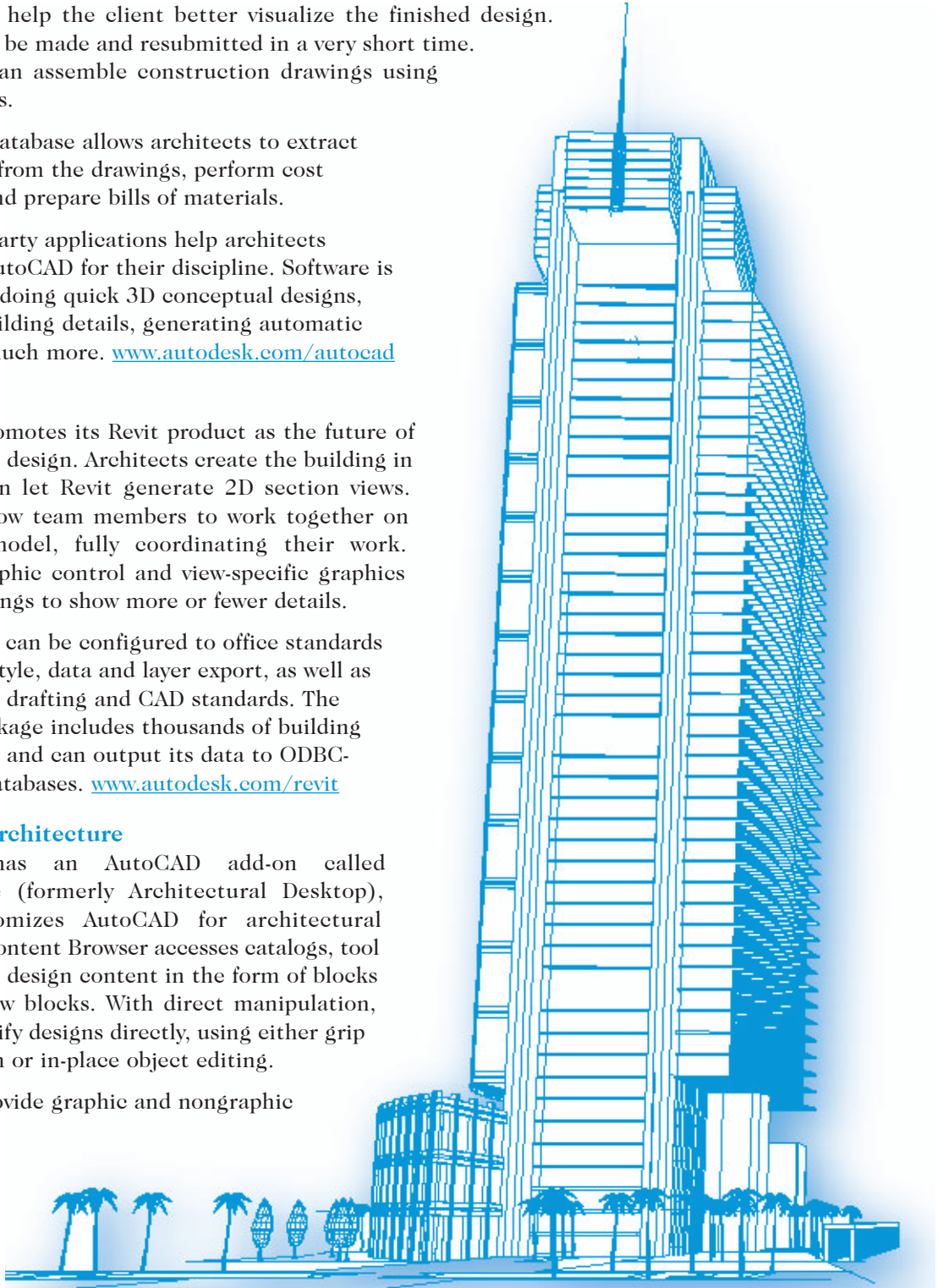
Autodesk promotes its Revit product as the future of architectural design. Architects create the building in 3D, and then let Revit generate 2D section views. Worksets allow team members to work together on the same model, fully coordinating their work. Detailed graphic control and view-specific graphics permit drawings to show more or fewer details.

The software can be configured to office standards for graphic style, data and layer export, as well as to additional drafting and CAD standards. The software package includes thousands of building components, and can output its data to ODBC-compliant databases. www.autodesk.com/revit

AutoCAD Architecture

Autodesk has an AutoCAD add-on called Architecture (formerly Architectural Desktop), which customizes AutoCAD for architectural design. Its Content Browser accesses catalogs, tool palettes, and design content in the form of blocks and multiview blocks. With direct manipulation, you can modify designs directly, using either grip manipulation or in-place object editing.

Materials provide graphic and nongraphic





attributes for design, documentation, and visualization. You can attach materials to building model objects and to their respective components for greater visual detail.

Scheduling allows tracking of any object in drawings. Because schedules are linked to data, they automatically update as the design changes. You can create schedule tables, objects tagged through xrefs, and schedules of external drawings.

Third-party developers have add-on software for roadway design, digital terrain modeling, and more. www.autodesk.com/autocadarchitecture

Engineering

Engineers use AutoCAD for many different kinds of design; in addition, they use programs that interact with AutoCAD to perform calculations that would take more time using traditional techniques. Among the many engineering disciplines benefitting from AutoCAD are civil, structural, and mechanical.

Building Systems

Autodesk's Building Systems software is meant for designing mechanical, electrical, and plumbing systems for buildings, such as offices. In this case, "mechanical" means heating, cooling, and fire protection. The Building Systems add-on helps you design splined flex ducts with editable grips; single-line, 2D, or 3D pipes for creating chilled water, hot water, steam, and other piping systems; rise/drop symbology for connecting ductwork; and fire protection content, such as sprinklers, deluge valves, and retard chambers.

The electrical design capabilities of Building Systems software include automatic calculation of service and feeder sizes, as well as panel schedules and single-, two-, and three-pole intelligent circuit objects that work across all referenced files.

For plumbing, the software includes schematic tools for piping layout; built-in plumbing code systems for automatically calculating pipe sizing and flow requirements; as well as 3D plumbing content and 3D piping tools. www.autodesk.com/buildingsystems



Autodesk's Civil 3D (formerly Land Desktop) performs COGO (coordinate geometry) tasks, and creates maps, models terrain, designs alignments, and parcels for land planning. Other features include topographic analysis, real-world coordinate systems, volume totals, and roadway geometry.

www.autodesk.com/civil3d

Interior Design

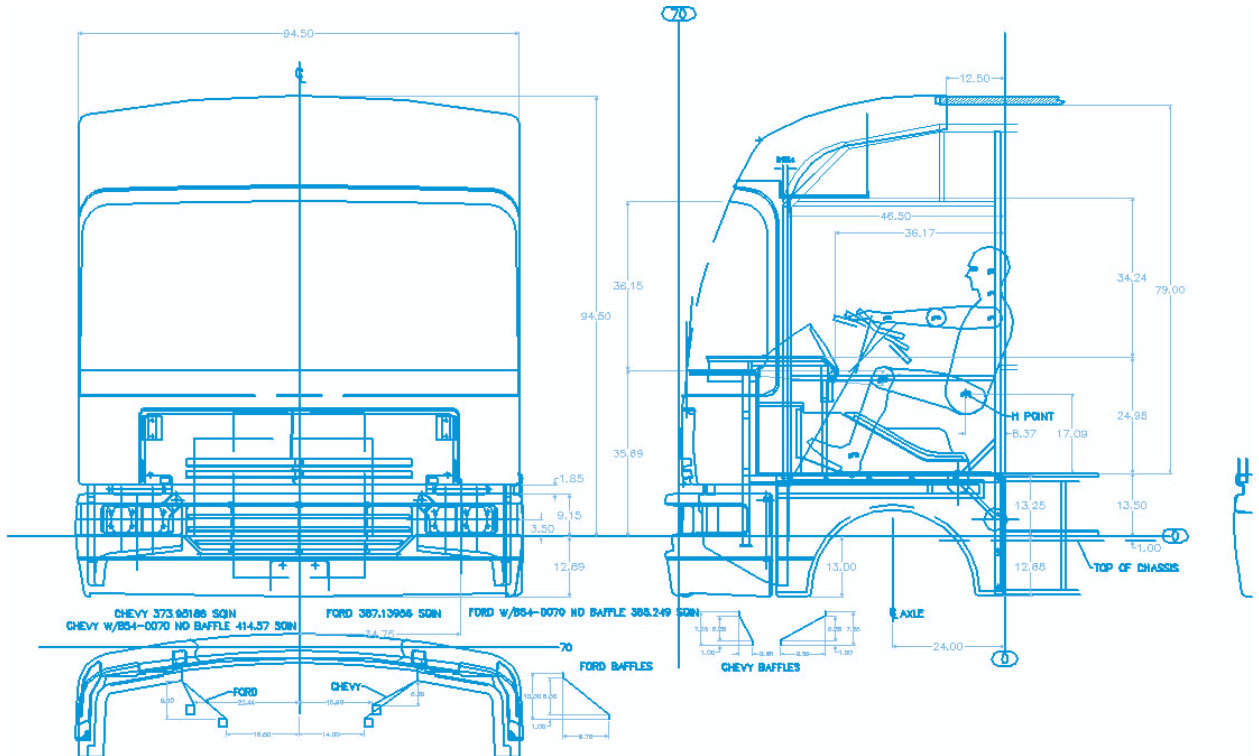
Manufacturing

The AutoCAD drawing on the next page shows how to fit the components of a truck cab around the driver.

Autodesk has several applications for mechanical engineers. For 2D drafting, there is AutoCAD Mechanical, and for 3D design, there is the Inventor Series.

AutoCAD Mechanical

AutoCAD Mechanical is an add-on program that runs on AutoCAD. It is meant for 2D drafting of mechanical parts. Its parts-tracking feature and large pre-drawn content make it easy for mechanical engineers to create, manage, and reuse their 2D drawing data. www.autodesk.com/autocadmechanical



Inventor Series

The Inventor software uses a paradigm different from “regular” AutoCAD, because mechanical parts can be very complex. You begin by drawing *parts* that make up *assemblies*. The assembly is the finished device, whether an MP3 music player or a plastic injection molding machine. The parts, as the name suggests, make up the assembly. Programs like Inventor are designed to handle tens of thousands of parts.

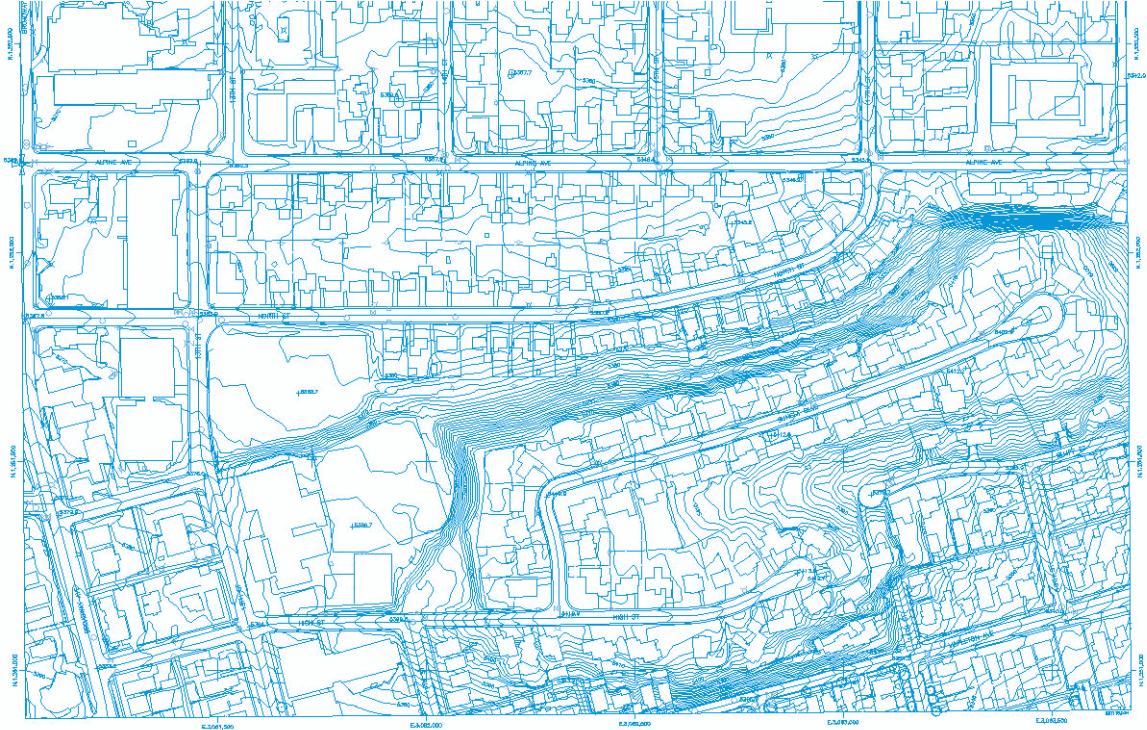
Once the assembly is created, Inventor can generate 2D plans, parts lists, and assembly instructions. Commands help place standard bolts, route wires and tubing, and determine whether the parts can withstand stresses from pressure and temperature. www.autodesk.com/inventor

The Inventor Series includes AutoCAD Mechanical and Inventor. (Inventor is independent of AutoCAD, but can read and write its drawings.) Inventor Professional adds discipline-specific functions, such as cable routing and PCB (printed circuit board) design. Inventor LT is a low-cost version that leaves out assemblies and add-on applications. www.autodesk.com/inventorlt

Facilities Management

After tenants move into the newly-constructed building, facilities managers can track how the building is used. When staff move to different offices, for example, they need to know whether the correct desks, chairs, cupboards, and telecommunications connections are available in their new office, or whether their existing furniture has to move with them. Tracking hundreds of offices is made easier using FMdesktop from Autodesk. www.fmdesktop.com

Shopping malls and airports use AutoCAD linked to a database to maximize the profit from leased stores. Owners of these complexes need to track the mix of stores, and the revenue they produce. Some stores don't want to be located near competitors; others want to be in prime locations, such as near the entrance or at a corner. Other data stored in the database include agreements with cleaning and maintenance contractors, even details such as floor coverings and air conditioner capacities.



Mapping

Map makers can use CAD to construct maps that store huge amounts of data. For example, a municipal map can contain information about all buildings, transportation systems, and underground services (such as storm sewer and water pipes). By using search criteria, the CAD system returns a map showing, for example, all water pipes older than 20 years. This map then shows where municipal engineers should replace old piping.

GPS (global positioning system) generates the data used to create maps in CAD. This system consists of 24 satellites that always circle the earth, broadcasting data about their location. A handheld GPS receiver fixes your location by analyzing data broadcast from at least three satellites.

Map 3D

Autodesk has a program specifically for AutoCAD users called Map 3D. It assigns properties to objects according to classification, and then identifies, manipulates, and selects objects by their description (e.g., road, river) rather than by their simple primitives (lines and arcs). Topology functions define how nodes, links, and polygons connect to each other. This permits network tracing, shortest-path routing, polygon overlaying, and polygon buffer generating.

Map has tools that help correct errors due to incorrect surveying, digitizing, or scanning. It connects to an external database, and can import data from other GIS products. www.autodesk.com/map3d

The GIS drawing above uses layers and colors to differentiate amongst geographic elements, such as homes, properties, streets, and underground services.

Entertainment

The entertainment industry is largely based on digital media, so CAD graphics are a perfect match. What you view on television and in the newspaper is produced by a form of CAD graphics. Movie and theater set designs are often done in CAD. Movie makers are turning to forms of electronic graphics for manipulations and additions to their filmed and animated work. Autodesk also sells media and entertainment software, such as Maya and MotionBuilder.

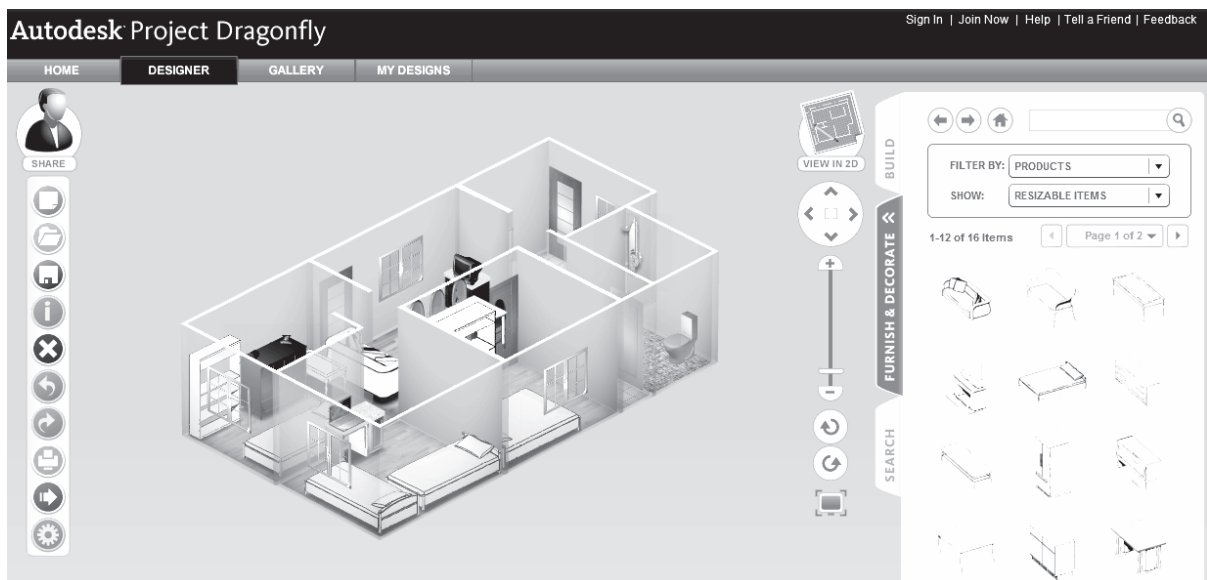
Online Applications

Autodesk is rolling out a series of applications that operate inside Web browsers. These programs are not like AutoCAD, but can assist with tasks that occur before and after drafting, such as job planning and rendering. At time of writing, all of these online applications were available from Autodesk Labs at labs.autodesk.com.

Much of this software works in Web browsers from FireFox and Microsoft, and generally requires JavaScript or Google Gears. The software is free to try while in the lab, although some require registration. This collection of software gives you an idea of the future of CAD.

Project Dragonfly

Dragonfly uses interactive design tools to rapidly create and furnish floor plan, allowing you to experiment in 2D and 3D, and in real time.

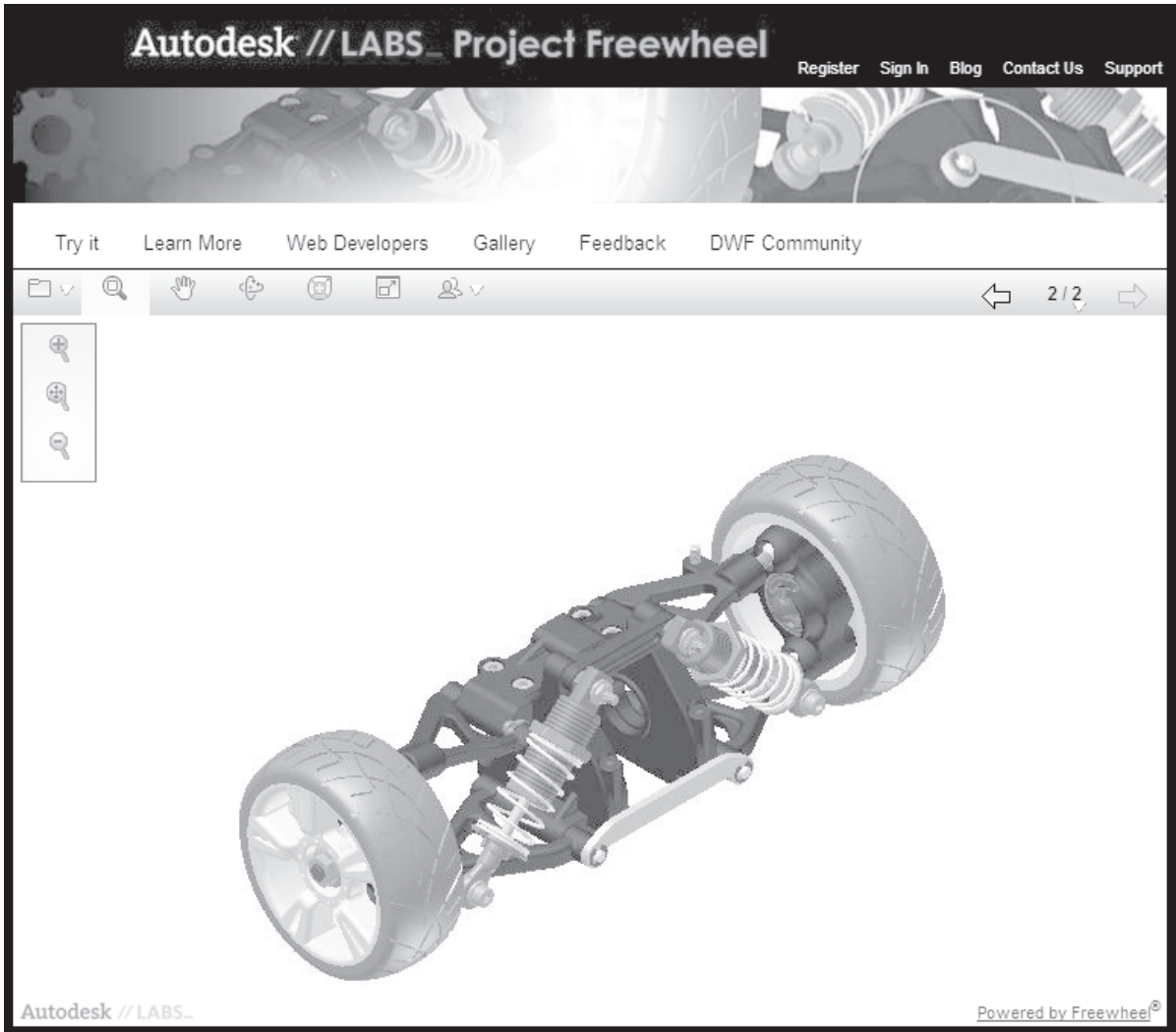


Project Draw

Draw is a Visio-like vector drawing environment for creating structured diagrams, such as organizational charts, simple floor plans, electronic circuits, network diagrams, and user interface mock-ups. It runs in JavaScript-enabled Internet Explorer or Firefox Web browsers.

Project Freewheel

Freewheel views and collaborates with DWF design files without installing additional software, because it runs in Web browsers. This means it works with Palm-size computers, cell phones, and on computers running Linux, Macintosh, and Windows.



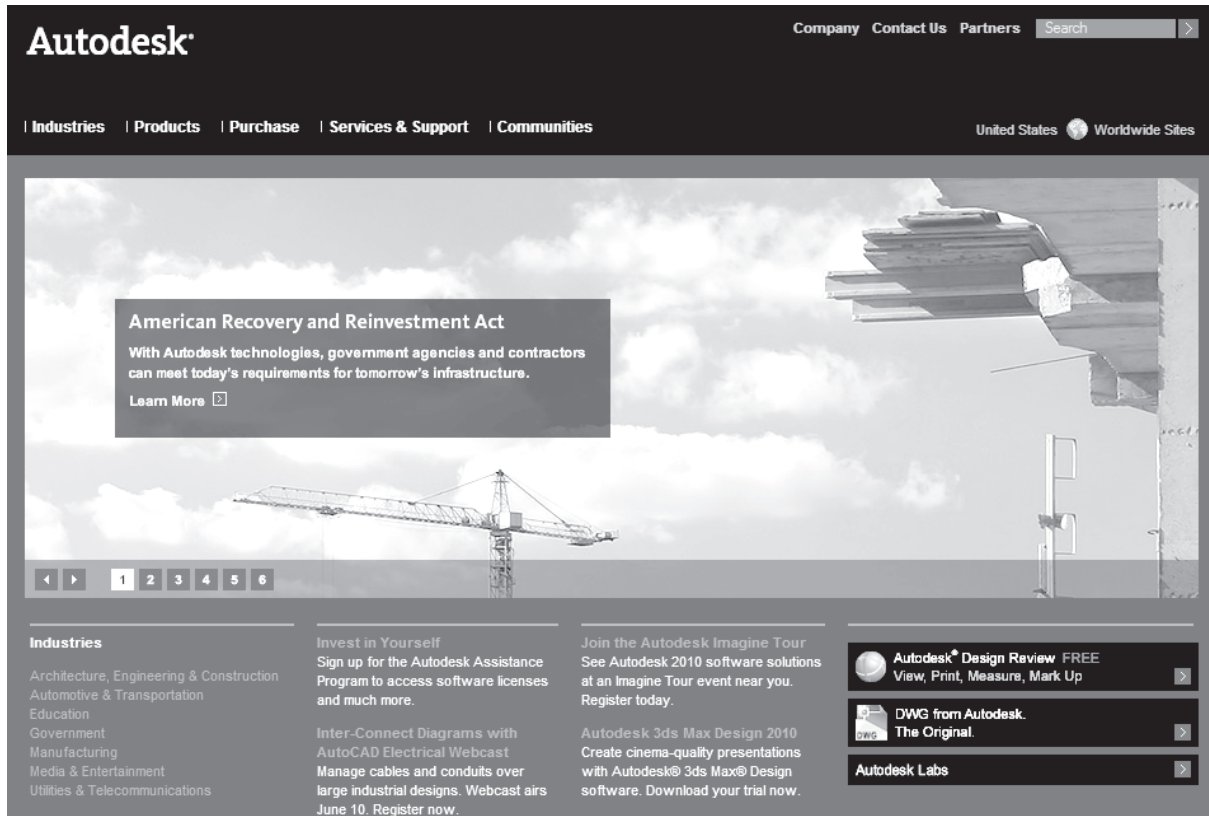
Project Newport

Newport is real-time 3D technology for architectural visualization and presentation that uses game-engine technology. This browser application lets architects to show their designs to clients, and quickly make changes to design options.



OTHER BENEFITS

Using a computer to draft and design also opens a new world of information technology. Many CAD systems now access email and the Web. There are Web sites specific to AutoCAD, as well as discussion groups and email newsletters.



Autodesk maintains many kinds of information at its Web site, www.autodesk.com. You can learn about updates to AutoCAD, download useful utilities, read up tips on for using AutoCAD, and access large libraries of symbols.

For help with bugs in AutoCAD, visit Autodesk's support site at support.autodesk.com.

AUTOCAD OVERVIEW

AutoCAD displays your drawings on the monitor or screen. All the additions and changes to your work are shown on the screen as you perform them. You can work with one or two monitors, placing the drawing on the main monitor and tool palettes on the secondary monitor.

Drawings are made up of 2D *objects*, such as lines, arcs, circles, text, as well as 3D objects, such as boxes, cylinders, and tori.

You place objects in drawings by entering *commands* and selecting options. You issue commands by several methods: typing the names of commands on the keyboard, selecting icons from the ribbon, or choosing options from floating palettes. Some commands start automatically when you double-click objects in drawings.

Once a command starts, you are often asked to choose among its options. After you identify all the information that AutoCAD requests, the new objects, or changes to objects, are shown on the screen.

If you need additional help, refer to Appendix B, “AutoCAD Command Summary.”

TERMINOLOGY

To use AutoCAD properly, this book contains terms and concepts that you need to understand. Some of the terms are explained briefly in this chapter. In addition, you may consult the index, which contains many terms.

Coordinates

AutoCAD uses the Cartesian coordinate system. The horizontal is represented by the x axis; the vertical, by the y axis. Any point on the plane can be represented by an x and y value shown in the form of x,y. For example 2,10 represents a point 2 units in the x direction and 10 units in the y direction.

The origin of the x and y axes is the 0,0 point. This point is normally at the lower left corner of the screen. You can, however, specify a different point for the lower left corner.

When AutoCAD works with 3D (three-dimensional) drawings, the third axis is called the “z axis,” which normally points out of the screen at you.

Drawing Files

Drawing files contain the information used to store the objects you draft. Drawing files automatically have the file extension of *.dwg* added to them.



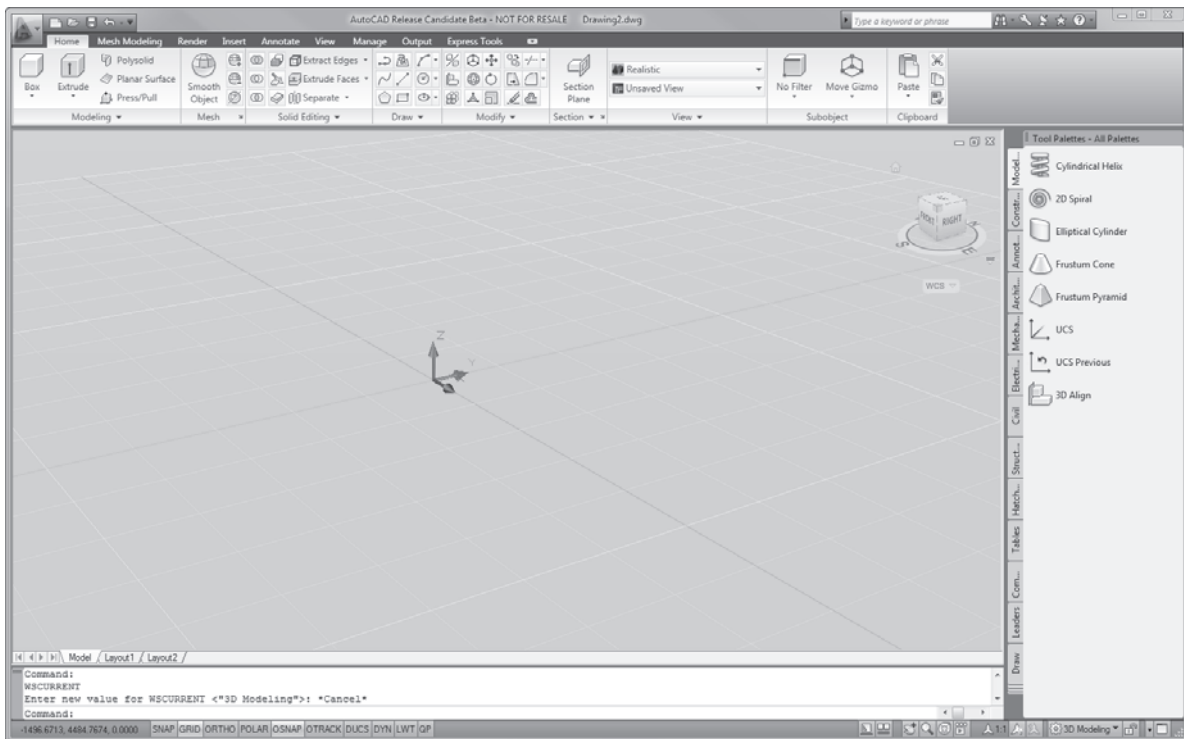
WHAT'S NEW IN AUTOCAD 2010

Throughout this book, the New in 2010 logo alerts you to features that are new in AutoCAD 2010. This book includes information on these new features.

USER INTERFACE

AutoCAD 2010 changes the arrangement of buttons, flyouts, and droplists on the ribbon. The ribbon is now more context-sensitive than before: when you execute certain commands or select specific objects, a tab related to the command appears in the ribbon. Autodesk calls these tabs “contextual tabs.”

Show below is AutoCAD 2010's user interface for 3D modeling



The menu browser is slimmed down to holding only file-oriented commands, such as for opening, exporting, and printing drawings. It has been renamed the “application menu.”

The new **3DSCALE** command scales objects along the 3D axes and planes.

The AdCenter palette replaces the DC Online tab with the new Autodesk Seek Design Content button. The related **SEEK** command opens a Web browser and displays the Autodesk Seek home page for finding blocks and drawings.

The **OPEN**, **QSAVE**, and **SAVE** commands open and save DWG 2010 drawing files by default, and support large 4GB objects; file names now auto-complete in the File Name text entry box.

Many similar commands have been collected in to “super” commands, such as **ATTACH**, **ADJUST**, **CLIP**, and **ULAYERS**.

2D DRAFTING

The **HATCH** command now indicates with red circles gaps in boundaries. Non-associative hatches can now be grip edited.

The **JOIN** command now joins splines to helixes, and helixes to splines.

The new **REVERSE** command reverses the order of vertices in lines, polylines, splines, and helixes; the Reverse option is also added to the **PEDIT** command. This is useful for flipping text in lintypes.

The **PEDIT** command's Spline option now converts polylines to true splines, instead of smoothed polylines. Similarly, the **SPLINEDIT** command now converts splines into polylines. The **MOVE**, **ROTATE**, and **SCALE** commands now move/rotate/scale a polyline segment relative to the rest of the polyline. Select the segment by holding down the Ctrl key when picking it.

The **MEASUREGEOM** command measures the distance, radius, angle, area, and volume of selected objects or sequence of points. When measuring areas and volumes, added areas are shown in green, while subtracted areas are in red.

PARAMETRICS

The new **DIMCONSTRAINT** command applies dimensional constraints between selected objects or on points of objects. This means you can specify distances between objects, as well as the sizes of objects. Linear, aligned, angular, radial, and diameter dimensional constraints are available for 2D drafting.

The new **PARAMETERS** command defines and edits parameters (text and formulas) associated with dimensional constraints.

The new **GEOMCONSTRAINT** command applies geometric relationships between objects or to points on objects. They define object snap-like relations between objects. For example, apply the Concentric constraint to two circles to keep them concentric; move one circle, and the other moves with it. Horizontal, vertical, perpendicular, parallel, tangent, smooth, coincident, concentric, colinear, symmetric, equal, and fix constraints are available for 2D drafting.

The new **AUTOCONSTRAIN** command automatically constrains selected objects in drawings. It finds probable constraints between geometric objects that are within a tolerance distances.

The new **CONSTRAINTSETTINGS** command controls the display and creation of geometric constraints, as well as the display of dimensional constraints. The dialog box also controls which constraints are applied, and in which order of priority.

3D MODELING

The new **MESH** command creates 3D mesh primitive objects: boxes, cones, cylinders, pyramids, spheres, wedges, and tori. The “deflated basketball” shown below was created by rotating a face on the sphere.



The new **MESHPRIMITIVEOPTIONS** command sets the tessellation defaults for primitive mesh objects created with the **MESH** command.

Subobjects of meshes can be edited directly by pushing or pulling them from the surface of mesh objects. Subobjects include edges, faces, and vertices.

The new **MESHSMOOTHMORE** command increases the level of smoothness of mesh surfaces by one level, while the related **MESHSMOOTHLESS** command decreases the level of smoothness.

The new **MESHCREASE** command sharpens the faces and edges of selected mesh subobjects.

The **EDGESURF**, **REVSURF**, **RULESURF**, and **TABSURF** commands now create 3D meshes when the new **MESHTYPE** system variable = 1.

The **EXTRUDE** command now works on faces of 3D mesh objects.

The **CHAMFER** and **FILLET** commands now bevel and fillet 3D meshes by temporarily converting them to 3D solids.

The **PRESSPULL** command now works with the **CTRL+SHIFT+E** keyboard shortcut on solid models.

TEXT AND DIMENSIONS

The **TEXTEDIT** command replaces **DDEDIT**, and edits dimensional constraints, dimensions, and all forms of text objects.

The **DIMSTYLE** command's Text tab has new options in the Text Placement section: the Vertical droplist adds the Position Text Below Dim Line option; the View Direction droplist reverse the direction of horizontal and vertical dimension text. The Primary Units and Alternate Units tabs have new subunit options that allow dimensions to show 50cm instead of 0.50m, for example.

PLOTTING

The **PUBLISH** command can now publish drawings as PDF files. As well, it automatically loads all open drawings, can specify precisions for DWF and PDF outputs, reads BP3 files, and allow demand-loading of third-party add-ons.

The Publish Options dialog box now handles PDF format and merge control (how overlapping lines are printed).

The **AUTOPUBLISH** command now publishes drawings as PDF files, as well as DXF/x documents, and controls how overlapping lines are merged (Merge Control).

USING THIS BOOK

Each chapter in *Using AutoCAD 2010* builds on the previous chapters. The basic use for each command is given, along with one or more tutorials that help you understand how the command works. This is followed by a comprehensive look at the command and its variations.

The exercises at the end of a chapter are specifically designed to use the commands covered in the book to that point. Review questions reinforce the concepts explored in the chapter. This method allows you to pace learning. Remember, not everyone grasps each command in the same amount of time.

CONVENTIONS

This book uses the following conventions.

Keys

Several references are made to keyboard keystrokes in this book, such as ENTER, CTRL, TAB, and function keys (F1, F2, and so on).

Control and Alternate Keys

Some commands are executed by holding down one key, while pressing a second key. The control key is labeled CTRL, and is always used in conjunction with another key or a mouse button.

To access menu commands from the keyboard, hold down the ALT key while pressing the underlined letter. The ALT key is also used in conjunction with other keys.

Flip Screen

The text and graphics windows can be alternately displayed by using the Flip Screen key. The F2 key is used for this function.

Command Nomenclature

When a command sequence is shown, the following notations are used:

Command: **line** (*Press ENTER.*)

Specify first point: (*Pick a point.*)

Specify next point or [Undo]: (*Pick point 1.*)

Boldface text designates user input. This is what you type at the keyboard. If the command and its options can be entered by other methods, these are described in the book. For example, commands can be selected from menus and toolbars, or entered as keyboard shortcuts.

(*Press ENTER*) means you press the ENTER key; you do not type “enter.”

(*Pick a point*) means that you enter a point in the drawing to show AutoCAD where to place the object. To pick the point, you can either click on the point, or type the x, y, z coordinates.

(*Pick point 1*) The book often shows you a point on a drawing. The points are designated, such as “Point 1.”

<Default> is how AutoCAD indicates *default* values. The text or numbers are enclosed in angle brackets. The default value is executed when you press ENTER.

[Undo] is how AutoCAD lists command *options*. The words are surrounded by square brackets. At least one letter is always capitalized. This means you enter the capital letter as the response to select the option. For example, if the option is **Undo**, simply entering U chooses the option.

Additional options are separated by the slash mark. When two options start with the same letter, then two letters are capitalized, and you must enter both.

COMPANION CD

This book includes a companion CD with the files needed for completing end-of-chapter exercises.

Exercise Files

Files needed for exercises are found on the Companion CD, included with this book. These files include AutoCAD drawing files, which are identified by the *.dwg* suffix in their file names, as well as other types of files. When an exercise requires a file from the CD, it plainly tells you its file name.

ONLINE COMPANION

The Online Companion™ is your link to AutoCAD on the Internet. We've compiled supporting resources with links to a variety of sites.

In addition, there is information of special interest to readers of *Using AutoCAD 2010*. This includes updates, information about the author, and a page where you can send your comments. You can find the Online Companion at www.autodeskpress.com/resources/oles/index.aspx. When you reach the Online Companion page, click on [All AutoCAD 2010 Titles](#).

e.Resource™ is an educational resource that creates a truly electronic classroom. It is a CD containing tools and instructional resources to enrich the classroom and reduce preparation time. The elements of e.Resource link directly to the text, and combine to provide a unified instructional system. With e.Resource, you can spend your time teaching, not preparing to teach.

Features contained in e.Resource include:

- **Instructor Syllabus:** This lists goals, topics covered, reading materials, required texts, lab materials, grading plan and terms. Additional resources and web sites covering descriptive geometry, manual drafting, AutoCAD, and Inventor are also listed.
- **Student Syllabus:** This lists goals, topics covered, required texts, grading plans and terms.
- **Lesson Plans:** These contain goals, discussion topics, suggested readings, and suggested homework assignments. You have the option of using these lesson plans with your own course information.
- **Answers to Review Questions:** These solutions help you grade and evaluate end-of-chapter tests.
- **PowerPoint® Presentation:** These slides help you present concepts and material. Key points and concepts can be graphically highlighted for student retention.
- **Exam View Computerized Test Bank:** This includes over 800 questions of varying levels of difficulty in true/false and multiple-choice formats that assess student comprehension.
- **AVI Files:** These movies, listed by topic, illustrate and explain key concepts.
- **DWG Files:** This list of .dwg files matches many of the figures in the textbook. These files can be used to stylize the PowerPoint presentations.

WE WANT TO HEAR FROM YOU!

Many of the changes to the look and feel of this new edition came by way of requests from and reviews by users of our previous editions. We'd like to hear from you as well! If you have any questions or comments, please contact:

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c/o Autodesk Press
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or visit our Web site at www.autodeskpress.com

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Setting Up Drawings

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CHAPTER 1

Quick Tour of AutoCAD 2010

Welcome to *Using AutoCAD 2010*!

As a new user, you need to experience the feel of AutoCAD firsthand. The concepts behind computer-aided design software can be disorienting to a first-time user, and so this quick-start chapter gets your feet wet. It introduces you to some of the features discussed in detail in later chapters.

In this chapter, you learn these commands:

- NEW** starts fresh drawings.
- LINE** draws straight line segments.
- U** undoes mistakes.
- QUIT** exits AutoCAD.
- CLOSE** and **CLOSEALL** close drawings.
- SAVEAS** renames drawings, and then saves them.
- PLOT** prints the drawing on paper.

Let's get started!

NEW TO AUTOCAD 2010 IN THIS CHAPTER

- The content of the ribbon is changed.
- The new application menu replaces the menu browser.

STARTING AUTOCAD

Before you can install and run AutoCAD, your computer must have Microsoft® Windows™ XP, Server 2003, Vista, or 7 installed and running.

TUTORIAL: STARTING AUTOCAD

I. To start AutoCAD 2010, use one of these methods:

- On the Windows desktop, double-click the icon labeled **AutoCAD 2010**. (*Double-click* means to press the left mouse button twice, quickly.)

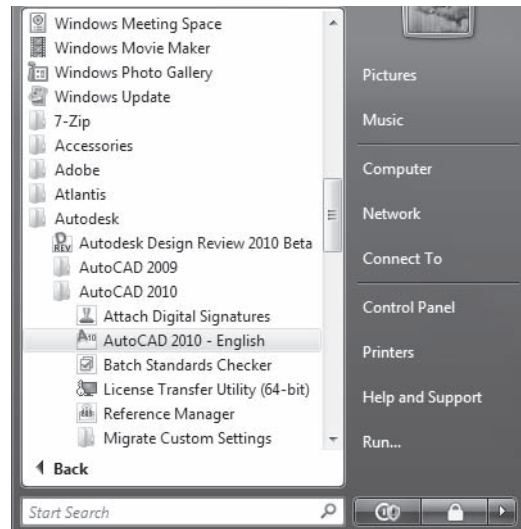


- Alternatively, you can also start AutoCAD from the Windows taskbar:

In Windows XP, click the **Start** button, and then choose **All Programs**. Next, choose **Autodesk | AutoCAD 2010 | AutoCAD 2010**.

In Windows Vista and 7, click the round Windows logo, and then choose **All Programs**, **Autodesk**, and then **AutoCAD 2010**.

(Your copy of Windows may differ from the display illustrated below.)

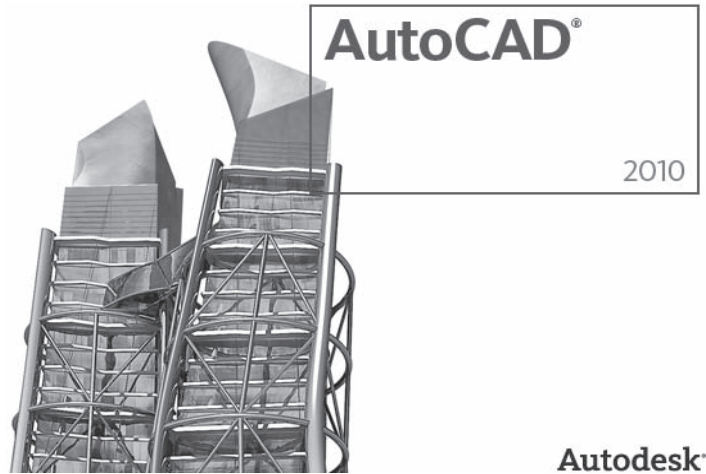


- Or, in the Windows Explorer, double-click a .dwg drawing file. (If AutoCAD add-ons, such as Mechanical Desktop or Architecture, are installed, they may start instead of AutoCAD.)
- Or, in the Run dialog box, enter **acad.exe**.

Run: **acad.exe** (Press **ENTER**.)

(This procedure bypasses command-line startup switches that may be associated with the icon.)

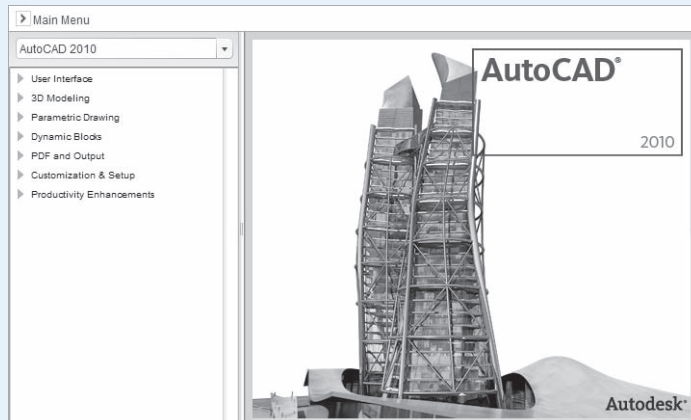
2. In all cases, Windows opens the AutoCAD software. An opening screen is displayed, called the “splash screen.”



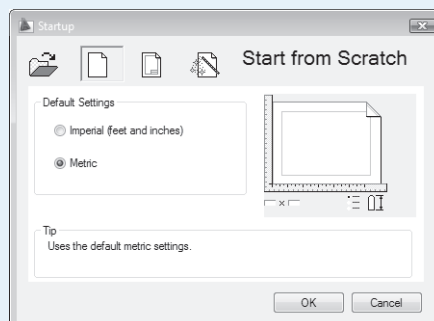
ADDITIONAL INITIAL DIALOG BOXES

AutoCAD sometimes shows several dialog boxes before it finishes starting up. Here are some you can expect to see — and instruction on what to do with them.

- If the **New Features Workshop** appears, choose **Maybe Later**, and then click **OK**.



- If the **Startup** dialog box appears, choose the **Start from Scratch** button, and then select **Imperial (feet and inches)**. Click **OK**.

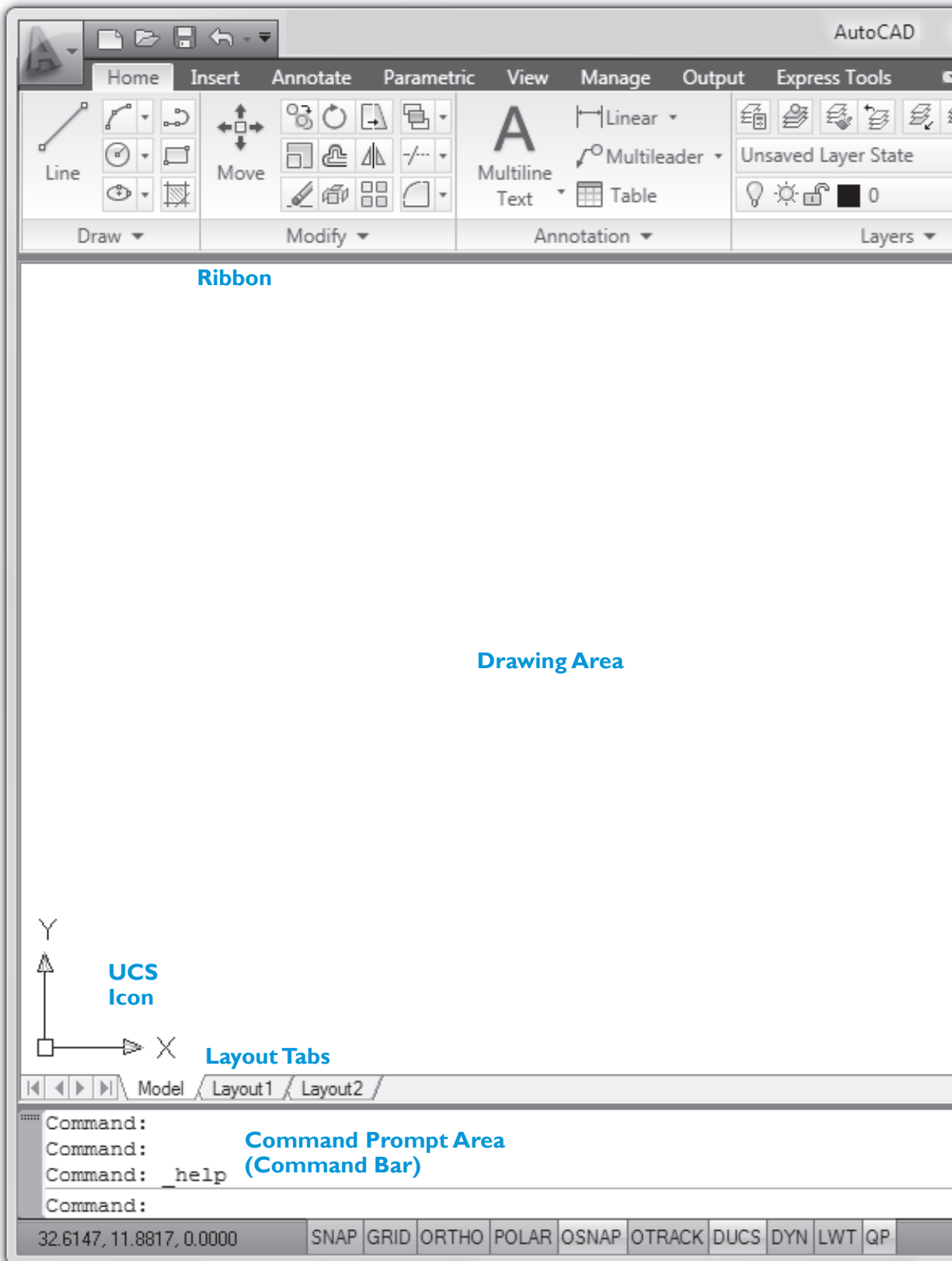


After the splash screen disappears, you see AutoCAD as illustrated by the figure on the following pages. (If you are familiar with older releases of AutoCAD, then the ribbon will be new to you — as will the lack of toolbars, menu bar, and layout tabs.)

**Application
Menu
(under A)**

Quick Access Toolbar

Title Bar & Drawing File Name

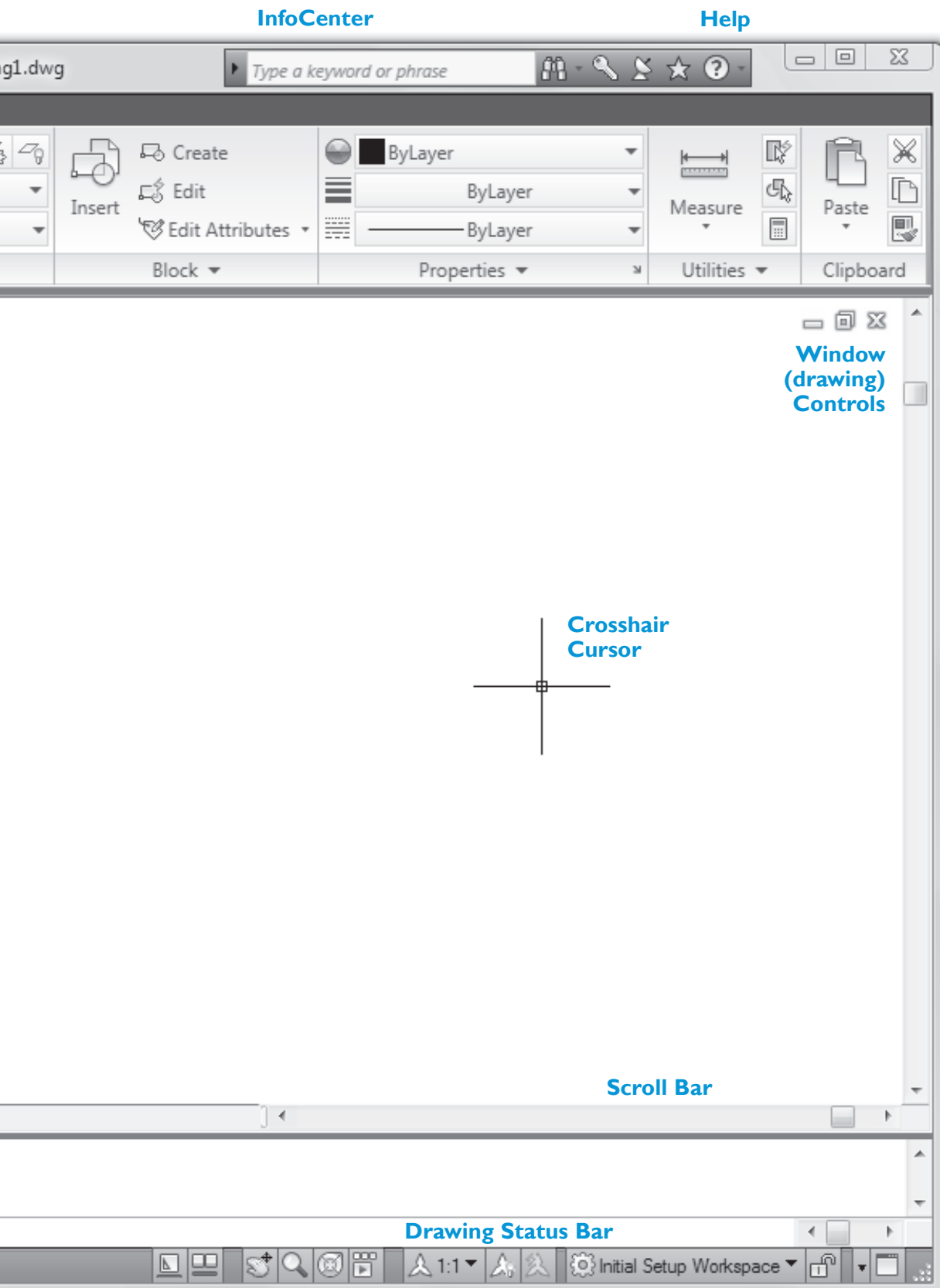


Drawing Area

Layout Tabs

**Command Prompt Area
(Command Bar)**

Status Bar



COMMANDS IN AUTOCAD

You make drawings in AutoCAD by creating and editing objects; you instruct AutoCAD to draw and edit through *commands*. Often, these are simple words such as LINE, CIRCLE, MOVE, and ERASE.

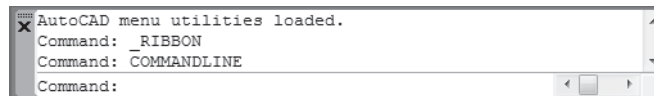
Commands describe the objects to be drawn — LINE or CIRCLE — and the operation to perform — MOVE or ERASE.

AutoCAD is very flexible in how you can control it; commands can be executed by several different methods, such as typing them on the keyboard, selecting them from ribbon panels, clicking a mouse button, or by entering shortcut keystrokes. As you work with AutoCAD, you will find that you will come to prefer one method over the others.

Despite the variety of input methods, AutoCAD is, however, always *command-driven*. Whether you click an item in a menu, on the ribbon or toolbars, or click a mouse button, AutoCAD translates the action in a command. Because Autodesk makes changes to AutoCAD's user interface every few years, longtime users prefer typing command names, because that's the one interface that never changes.

Command Bar

At the bottom of the AutoCAD window is the *command bar*. (If you do not see the command bar, press CTRL+9 to make it appear.)



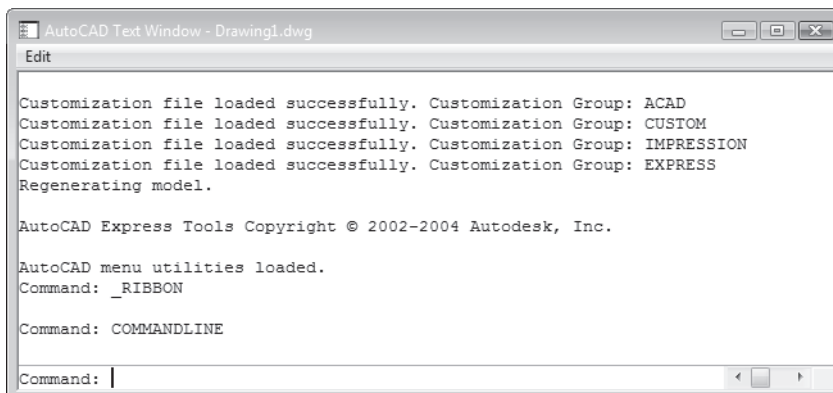
This is where you enter typed commands, and one of the places where AutoCAD responds back to you. You should see the word “Command:” on the bottom line now, like this:

Command:

You enter command names at this prompt, and then read instructions from AutoCAD on the steps to follow.

Text Window

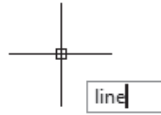
AutoCAD has a larger version of the command bar called the “text window.” It displays only text. You can see it by pressing function key F2; press F2 a second time to bring back the graphics window.



Dynamic Input

In addition to the command bar, the drawing area itself is an area where you can enter commands and receive feedback from AutoCAD. When *dynamic input* is turned on, the command prompts and options appear in small rectangles near the cursor.

In the figure below, the “line” command is being entered.

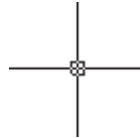


Normally, dynamic input is turned on by default; if not, click the  (or DYN) button on the status bar, and then ensure the cursor is in the drawing area. You learn more about dynamic input in Chapter 2, “Understanding CAD Concepts.”

Mouse

An alternative to typing commands is using the mouse. To see how the mouse interacts with AutoCAD, follow these steps:

1. Move the mouse, and notice how a crosshair cursor moves around the screen. (When the cursor looks like the crosshair illustrated below, you can use it to specify points in drawings.)



2. Move the crosshair cursor out of the drawing area. Notice that the cursor changes to an arrow. In this state, the cursor lets you select commands from the ribbon, application menu, and other areas.

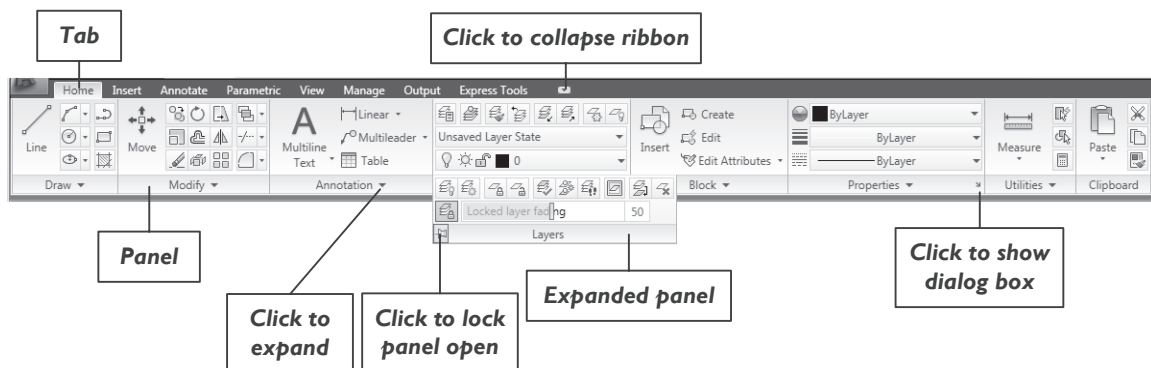


As alternatives to the mouse, you can also use digitizing tablets or other input device with AutoCAD. From this point forward, however, we refer to the mouse only, being the most common of pointing devices.

In the following sections, we look at how the ribbon, application menu, and other user interface elements operate. All of these are accessed with the mouse.

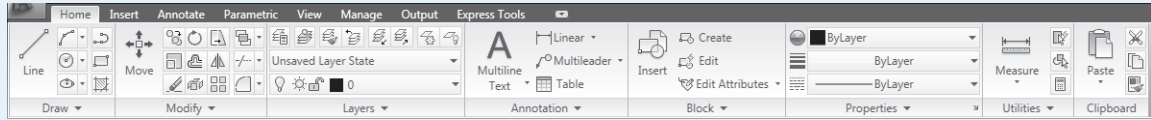
RIBBON

The *ribbon* is prominent as the primary interface. It spans the upper portion of AutoCAD. Through the use of tabs and panels, it segregates commands into logical groupings. (The ribbon replaces the Dashboard of just two earlier releases.)



ABOUT RIBBON TABS AND PANELS

Tabs represent major groupings of tasks in AutoCAD. They are identified by names such as Home, Visualize, and View. To access a tab, click its name.

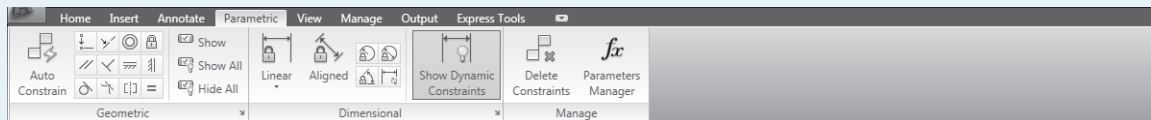


The Home tab (illustrated above) contains the most commonly used AutoCAD commands, such as those that draw lines, move and copy objects, control layers, and place text. Sometimes, AutoCAD automatically displays tabs, such as while editing text, tables, and blocks (symbols).

AutoCAD provides ribbon sets that differ slightly for 2D, 3D, and other workspaces. (*Workspaces* customize AutoCAD's user interface, and are described later in this chapter.) The ribbon for the 3D workspace is illustrated below. For now, we will use the 2D Home panel.



Each tab consists of one or more *panels*. Each panel contains a group of related commands. For example, click the name of the **Parametric** tab to see panels for placing and editing geometric and dimensional parameters (illustrated below).



Ribbon Shortcuts

Some panels aren't big enough to hold all commands, and so you can expand them to see additional ones. To expand a panel, click the triangle icon; to keep the panel expanded, click the thumbtack icon — as illustrated on the previous page.

You can drag panels away from the ribbon to keep them open all the time. Indeed, you can *undock* the entire ribbon, and then drag it to a second monitor. (Right-click a tab name, and then select **Undock** from the shortcut menu.)

Some panels have related dialog boxes for setting options. You access the dialog box by clicking the arrow icon.

If the ribbon takes up too much room on your computer's screen, you can minimize it by clicking the minimize button. Click it once to collapse it down to tab and panel names.



Click it again to show just tab names. (Panels appear again when you hover the cursor over their names.)



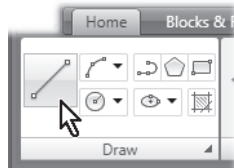
Click the button a third time to expand the panels to their default state.

Each panel contains *buttons*. Buttons are typically labeled with *icons*, the Greek word for “small picture.” Each picture represents the function of its button. When you click a button, AutoCAD executes the related command.

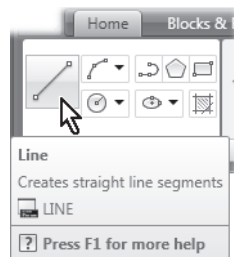
TUTORIAL: EXECUTING COMMANDS WITH THE RIBBON

Commands are executed in AutoCAD by clicking buttons, selecting options from droplists, or dragging sliders. For example, to start drawing lines you can click the button for the Line command. (Make sure the Home tab is active, the 2D Drafting workspace is current, and Ortho and Polar are turned off.) Follow these steps:

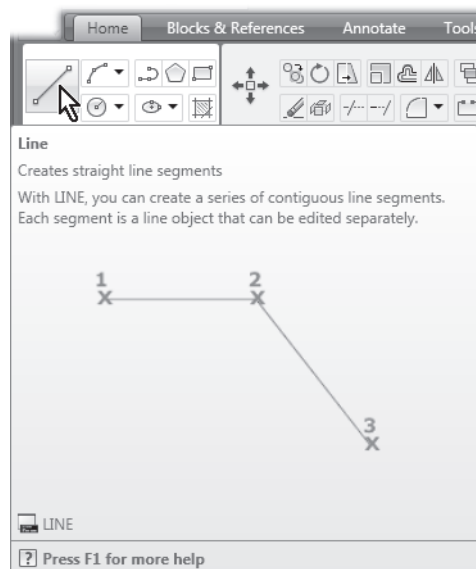
1. Move the cursor up to the ribbon, and then pause it over the large  button. The icon on the button tells you that it starts the **LINE** command. Don't press the mouse button yet!



If you wait a couple of seconds with the cursor on the button, a *tooltip* appears that tells you the purpose of the button:



Wait a few seconds longer, and the tooltip expands to give you further details on the **LINE** command:



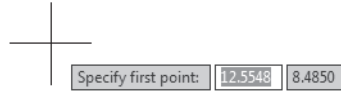
2. With the cursor on the **LINE** button, click the left button on your mouse. Notice that text appears in the command bar at the bottom of the window:

Command: `_line` Specify first point: (*Pick a point.*)

This is called a *prompt*.

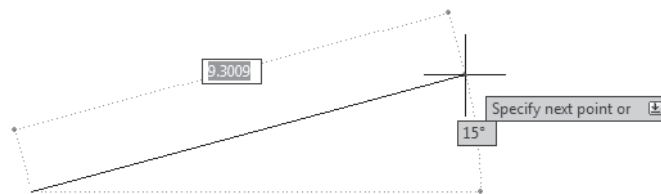
In the command bar, AutoCAD tells you what it expects from you. Here, AutoCAD asks you to specify the point at which the line starts, called the “first point.”

3. Move the crosshair cursor into the drawing area. Notice that the prompt is duplicated near the crosshair cursor. This is called the “dynamic prompt,” because it moves with the cursor.



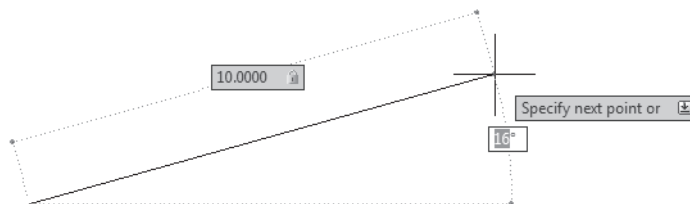
Next to the dynamic prompt, AutoCAD reports the current x and y coordinates.

4. Click the left mouse button to specify the starting point of the line.
5. Continue to move the crosshair cursor across the screen. Notice how a line “sticks” to the intersection of the cursor crosshairs. This is called the “rubber band.” It stretches and follows your movement to preview the location of the line, when you click again.

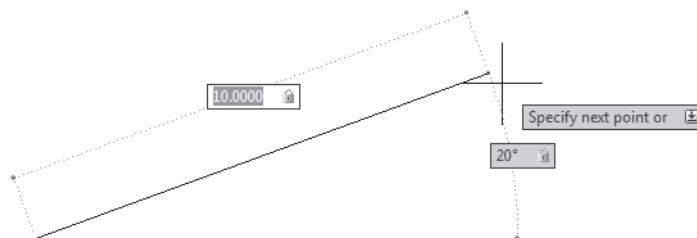


AutoCAD uses dynamic prompts to report the current length and angle from the x axis. In the illustration above, the length is 9.3009 units and the angle is 15 degrees.

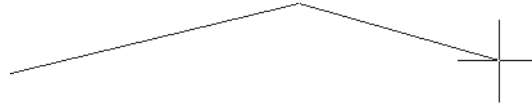
6. To enter a specific length:
 - a. Enter a value at the keyboard, such as **10**. Notice that it appears in the length box.
 - b. Press the **TAB** key. Notice that a padlock icon appears. The padlock indicates that the length is *locked*; it cannot be changed by moving the cursor — although you can certainly override it by entering another length.



7. Move the cursor. Notice that the length remains the same, even as the angle changes. To indicate a specific angle, enter the value, such as **20**, and then press **TAB**. Now the angle is also locked, as indicated by the second padlock.



8. Move the cursor, and pick another point.



In the command prompt area, the prompt has expanded with the addition of the words “or [Undo]”:

Specify next point or [Undo]: (Pick another point.)

The **Undo** word is an *option*. Options are alternative actions that can be performed by the command. **Undo** “undraws” the previous line segment, which is useful when you make mistakes or change your mind. AutoCAD shows options in square brackets, such as [Undo].

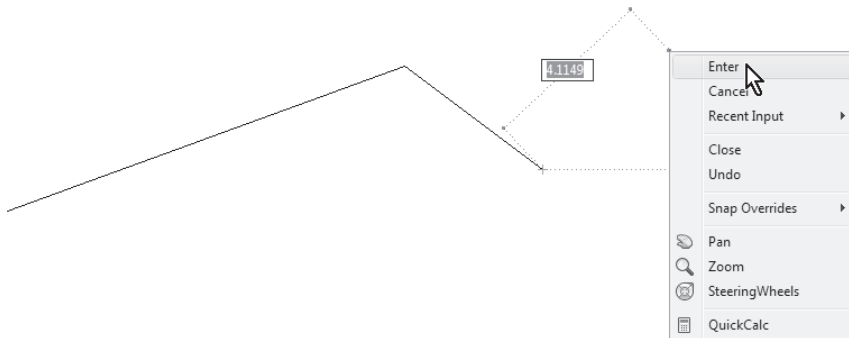
9. The **LINE** command remains active until you cancel it. By remaining active, it allows you to draw as many segments as you require — without reselecting the command from the ribbon.

Continue to enter line segments. As you do, the prompt changes to:

Specify next point or [Close/Undo]:

Close is another option of the **LINE** command. When you enter **Close**, it closes the polyline by drawing a line segment from the end of the last segment to the start of the first segment automatically.

10. We’ll end the command now by clicking the right mouse button. From the shortcut menu, choose **Enter**, as illustrated by the figure below. Notice that the shortcut menu gives you access to several other options of the **LINE** command (Close and Undo), as well as other AutoCAD commands, such as **PAN**, **ZOOM**, and **QUICKCALC**.



The **Enter** option ends the command; you can also press **ENTER** (or the spacebar) on the keyboard. (Alternately, press **ESC** on the keyboard; for some commands, you will need to press **ESC** twice before they end.)

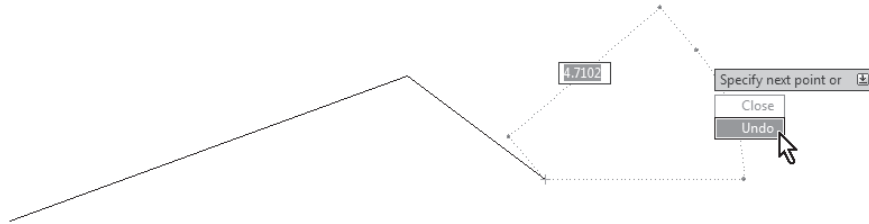
11. To erase the lines you drew, click the  **Undo** button in the Quick Access toolbar. (Alternatively, enter the **U** command at the ‘Command:’ prompt, short for “undo.”)

Command: **U** (Press **ENTER**.)



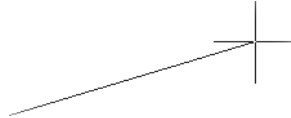
Notes If you wish to draw lines that are perfectly horizontal and vertical, hold down the **SHIFT** key while drawing.

You can also access options of the **LINE** command through dynamic input. Press the  down arrow on the cursor keypad to see the options, as illustrated below.



Frankly, I find it quicker to enter the options at the keyboard.

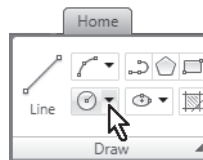
If dynamic input is turned off, then the prompt does not appear, and the screen looks like this:



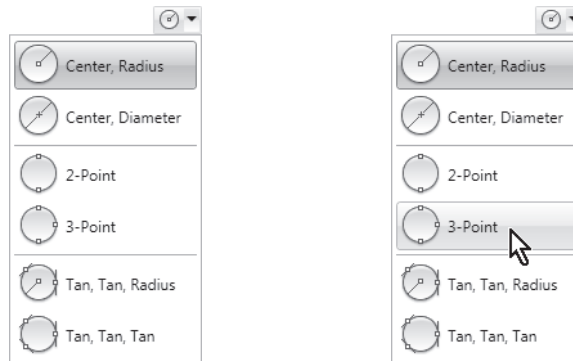
TUTORIAL: ACCESSING FLYOUTS

Sometimes buttons on the ribbon hide additional buttons known as *flyouts*. You recognize these buttons by the tiny ▼ black triangle in the corners of icons. To access flyouts, follow these steps:

1. Position the cursor over the tiny triangle next to a button, such as the Circle button illustrated below. (If you click the **Circle** button itself, AutoCAD executes the **CIRCLE** command.)



2. Click the left mouse button. Notice that AutoCAD displays a flyout that lists alternate ways of drawing circles. See figure below, at left.

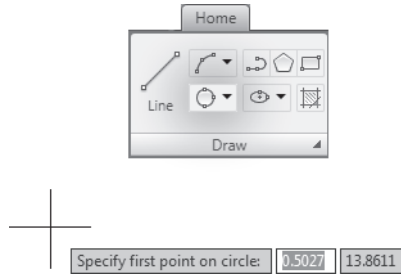


Left: Flyout of command options.

Right: Selecting an option.

3. Drag the cursor over the buttons on the flyout. As the cursor passes over a button, it changes slightly to give the illusion of being depressed. See figure above, at right.

- When you reach the button you want, click the mouse button. AutoCAD starts the command associated with the button.

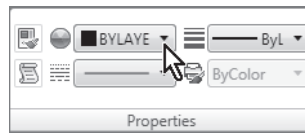


The icon associated with the option you selected moves to the top of the button, where it is the default next time.

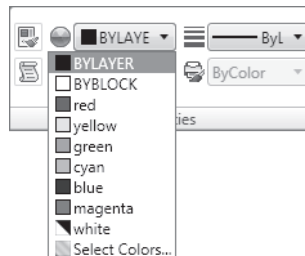
TUTORIAL: ACCESSING DROPLISTS

Another way the ribbon compresses multiple choices is through *droplists*. For example, the names of layers and range of colors are represented by droplists. To access a droplist, follow these steps:

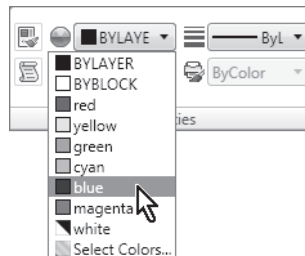
- Position the cursor over a droplist, such as the Color droplist found in the Home tab's Properties panel.



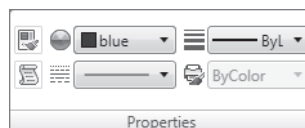
- Click the droplist. Notice the list of options that extends downwards.



- Move the cursor down the list, and then choose an option, such as the color **Blue**.

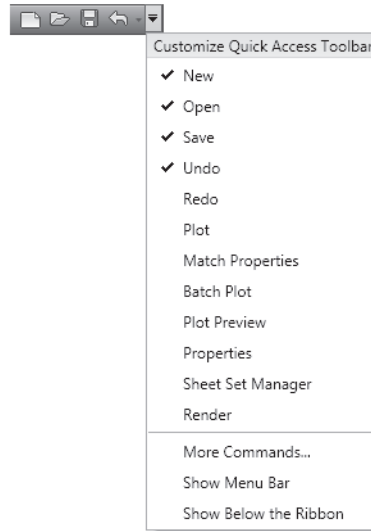


- Click to choose the option. Notice that the list disappears, and that the option you chose now appears in the droplist.



QUICK ACCESS TOOLBAR

The Quick Access toolbar is found on the title bar, at the top of the AutoCAD window.



The row of buttons contains some common utility commands:

-  Starts a new drawing (executes the **NEW** command).
-  Opens existing drawing files (**OPEN**).
-  Saves the current drawing (**SAVE**).
-  Undoes the last command (**U**).

You add commands to the toolbar by clicking the arrow, and then choosing from the menu.

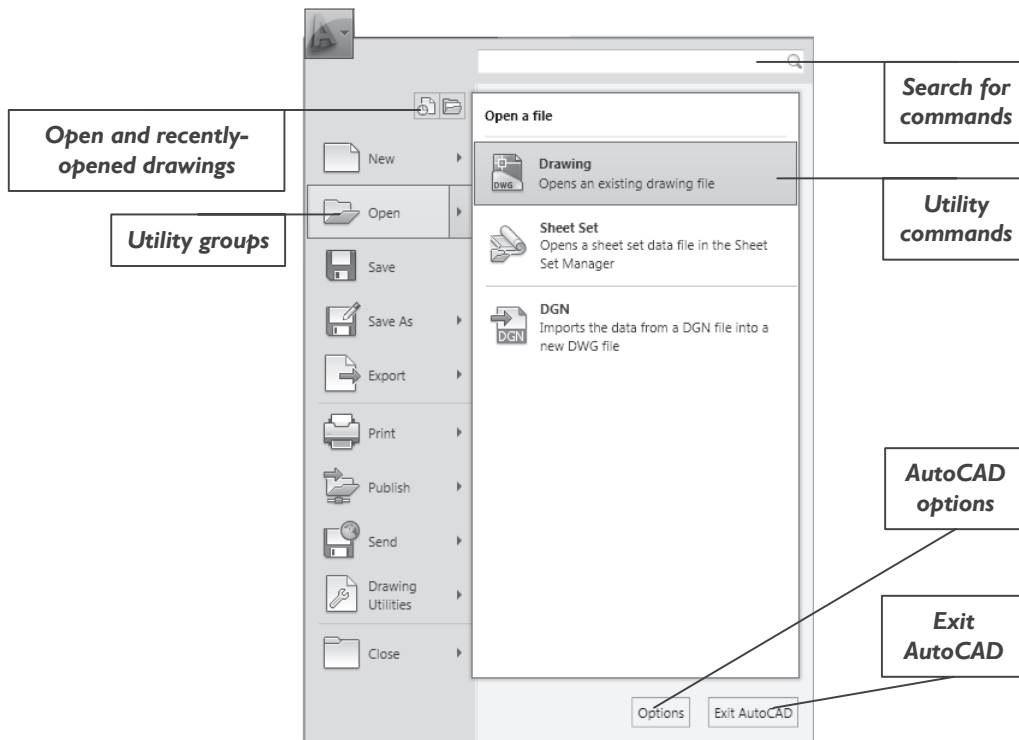


ACCESSING THE APPLICATION MENU

A big red A resides in the upper left corner of the AutoCAD window.



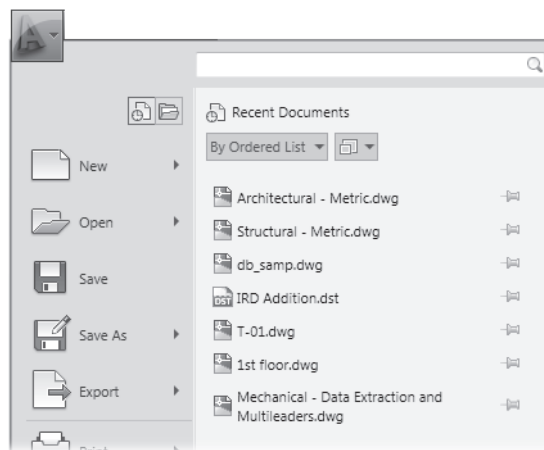
More than just a logo, the A hides the *application menu* and its utility commands. (This new menu replaces the menu browser of AutoCAD 2009.) To see the menu, click the A.



Recent Drawings

The application menu lists the names of previously opened drawings — useful when you want a drawing file but can't remember its name or location. (Autodesk calls this feature “Recent Documents,” even though AutoCAD does not open Office-style documents.)

Click the **A**, and then click the **Recent Documents** button. Notice the list of previously opened drawing file names in the right-hand pane.



To open a drawing, click on its file name.

The pane contains several useful options. The By Ordered List droplist lets you sort drawings by order (the order in which they were opened), their file size, file type, or by the date last accessed.

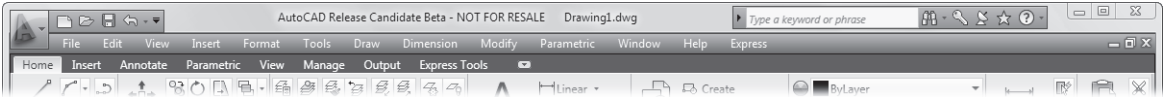
The other droplist displays drawing names as icons or images.

When the list gets long enough, the oldest file names are removed automatically. Click the  pushpin to keep them from falling off the end of the list.

The  Options button displays the Options dialog box (OPTIONS command), and the  Exit AutoCAD button exits AutoCAD (EXIT command).

Displaying the Menu Bar

The application menu has a few drawbacks. For instance, you need to click the A every time you want to access the menus. And it lacks most AutoCAD’s commands. The alternative is to turn on the menu bar, which then appears above the ribbon. It provides faster access to commands in menu format.

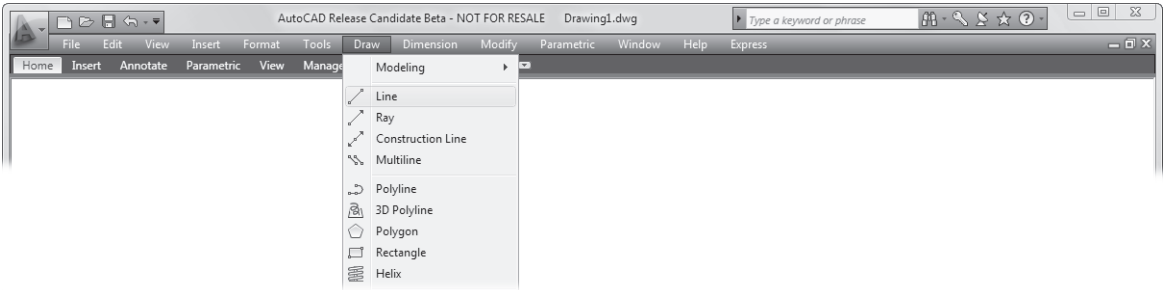


To display the menu bar, enter the following at the ‘Command:’ prompt:

Command: **menubar**
Enter new value for MENUBAR <0>: **1**

Enter the number **1**, and then press ENTER. (In this case, “1” means on.)

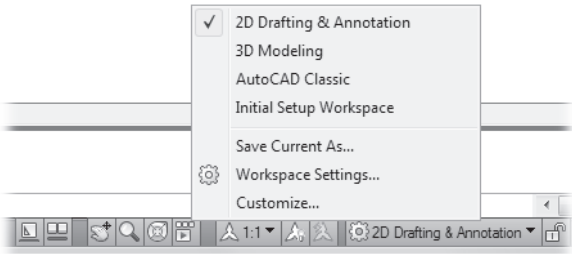
Like the ribbon, the menu bar groups commands by function. For instance, the Line command is found in the Draw category. Unlike the ribbon, these menus always display words alongside icons.



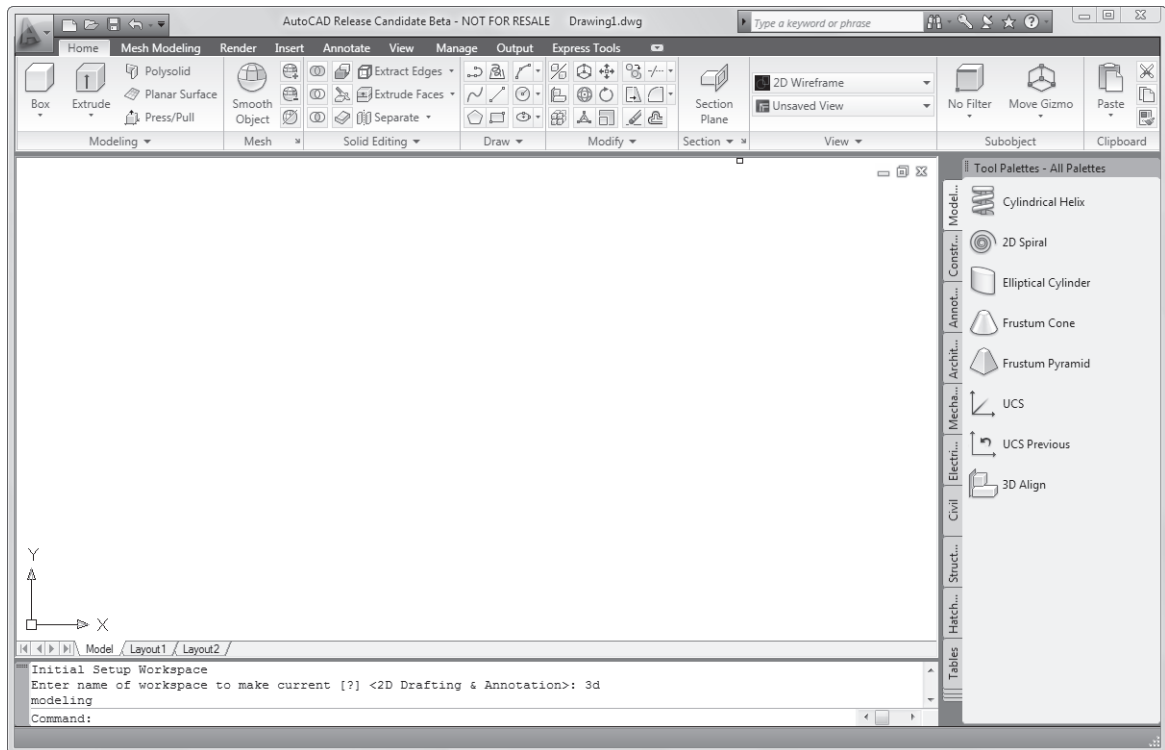
CHANGING WORKSPACES

The way that AutoCAD looks — its menus, toolbars, ribbon, and palettes — is controlled by *workspaces*. Selecting another workspace changes the number and position of these user interface elements. Four workspaces predesigned by Autodesk, which you select from the Workspaces button on the status bar, are described below.

- 1. Move the cursor to the right end of the status bar, and then click the  Workspaces button. Notice the popup menu:



- 2. From the menu, choose **3D Modeling**. Notice that AutoCAD’s user interface changes. The ribbon has different panels, and a Tool palette appears on the right.

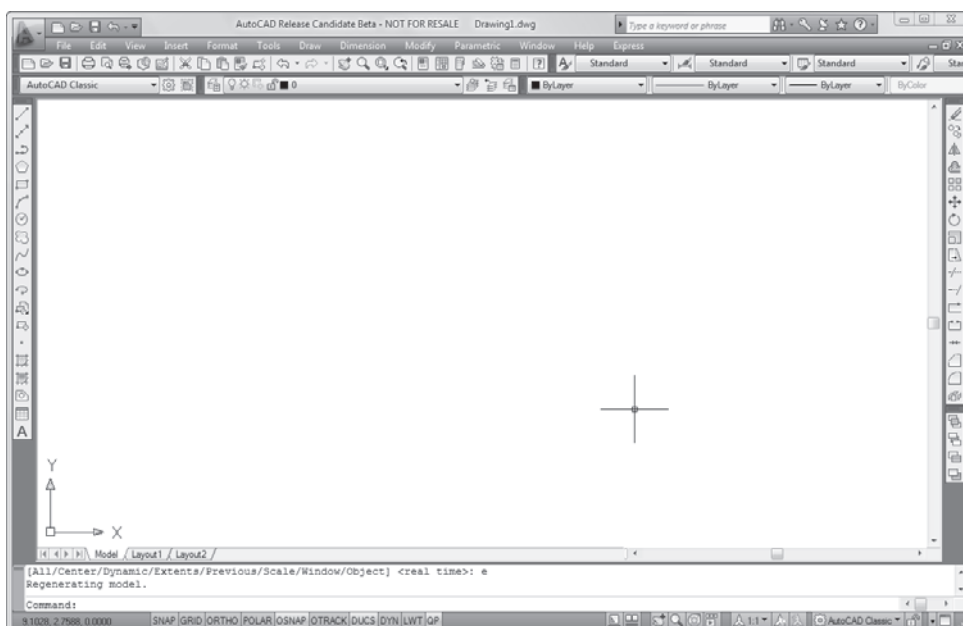


3. This book works with AutoCAD's default workspace, "2D Drafting & Annotation." To return AutoCAD to its default look, select this name from the Workspace popup menu.

Workspaces are described in greater detail in Chapter 3, "Setting Up Drawings." For now, it is sufficient to know that if you do not see a user interface element, your workspace may have changed.

AutoCAD Classic Workspace

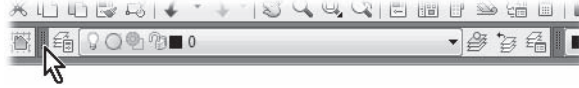
If you are used to working with the menu bar and toolbars and without the ribbon (as in AutoCAD 2008 and earlier), then choose **AutoCAD Classic** as the workspace. Notice that the menu bar, toolbars, and layout tabs appear. (If you also want the ribbon, redisplay it with the **RIBBON** command.)



Toolbars

The Classic workspace predominately features toolbars. They consist of collections of buttons labeled by icons, and group similar commands together. For example, many drawing commands appear on the Draw toolbar. You can float, move, resize, dock, and dismiss toolbars. To float a docked toolbar, follow these steps:

1. Position the cursor over the bar of dots at the left end of horizontal toolbars (or the top end of vertical toolbars).



2. Hold down the mouse button, and drag the toolbar away from the edge of the window.



3. Release the mouse button.

The toolbar now *floats* in the drawing window.

As of AutoCAD 2009, floating toolbars no longer sport the name of the toolbar, which makes identification difficult. The workaround is to pause the cursor over the toolbar and wait for the tooltip to appear, as illustrated below.



To move a floating toolbar:

1. Position the cursor over either end of the toolbar.
2. Hold down the mouse button and drag the toolbar to another location.



To resize (or stretch) a toolbar:

1. Position the cursor over one of its four edges. Notice that the cursor becomes double-ended.
2. Hold down the left mouse button, and then drag the toolbar to achieve a new shape.



The thick rectangle previews the new shape of the rectangle.

To dock a toolbar, drag the toolbar against one of the four sides of the drawing area.

Docking means to move it to the side or top of the drawing area, like the docked toolbars below the menu bar, illustrated in the figure below.

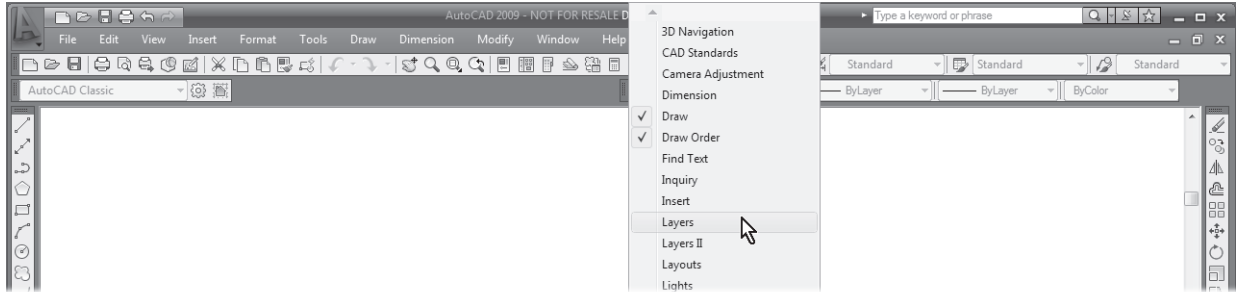


To dismiss (get rid of) a floating toolbar, click the tiny **X** in the upper right corner. The toolbar disappears.



To get back a toolbar:

1. Right-click any toolbar button.
2. From the shortcut menu, select its name from the list.



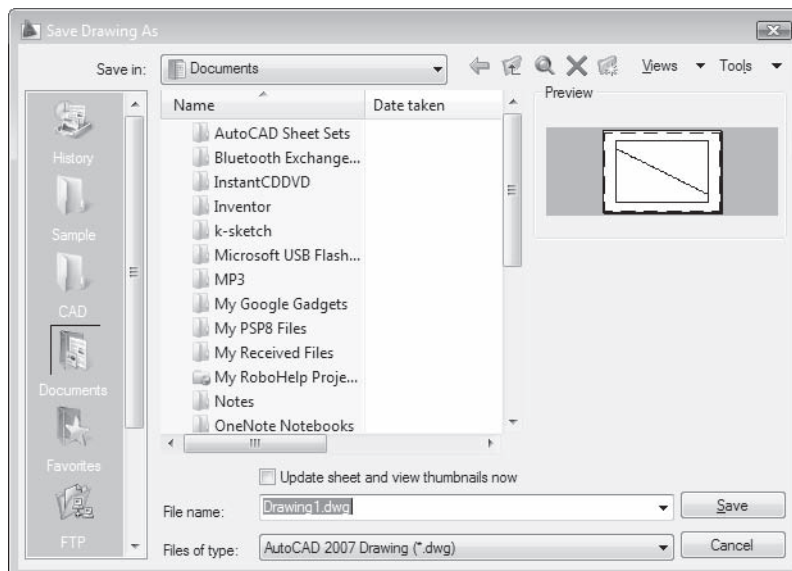
Leave at least one toolbar visible so that you can easily access the other toolbars. If all of them are closed (such as in the default workspace), you can use the **View** menu's **Toolbars** item to reopen them.

EXITING AUTOCAD

AutoCAD provides several different ways to close drawings and exit AutoCAD. The following sections outline the possibilities.

To save your work and remain in AutoCAD:

1. Click the  **Save** button in the Quick Access toolbar.
2. If the drawing has not yet been named, the Save Drawing As dialog box appears.



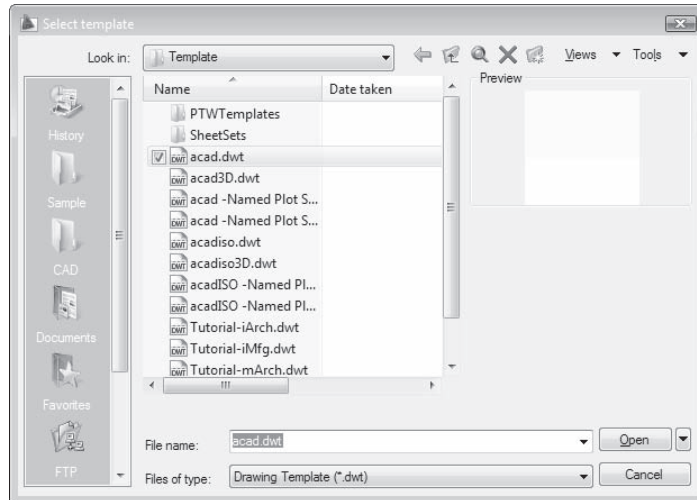
3. Name the drawing in the **File name** text box, and then choose **Save**.

To close the drawing and remain in AutoCAD:

- Enter the **CLOSE** command to close the current drawing. If necessary, AutoCAD asks if you wish to save the drawing. AutoCAD then closes the drawing.
- To close all drawings at once, enter the **CLOSEALL** command.

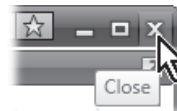
To start a new AutoCAD drawing:

1. Click the  **New** button in the Quick Access toolbar.
2. AutoCAD displays the Select Template dialog box. Select a template file, and then click **Open**.



To save your work and exit AutoCAD:

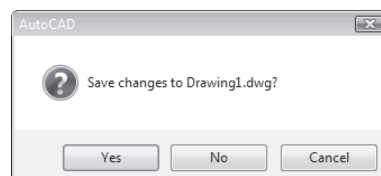
1. Click the **X** button found in the upper right corner of the AutoCAD window, adjacent to the InfoCenter.



2. When the “Save changes?” dialog box appears, choose **Yes** to save your work.
3. AutoCAD displays the Save Drawing As dialog box — if this is a new drawing that has not been saved previously. AutoCAD saves your work, and then exits to the Windows desktop.

To discard the drawing and exit AutoCAD:

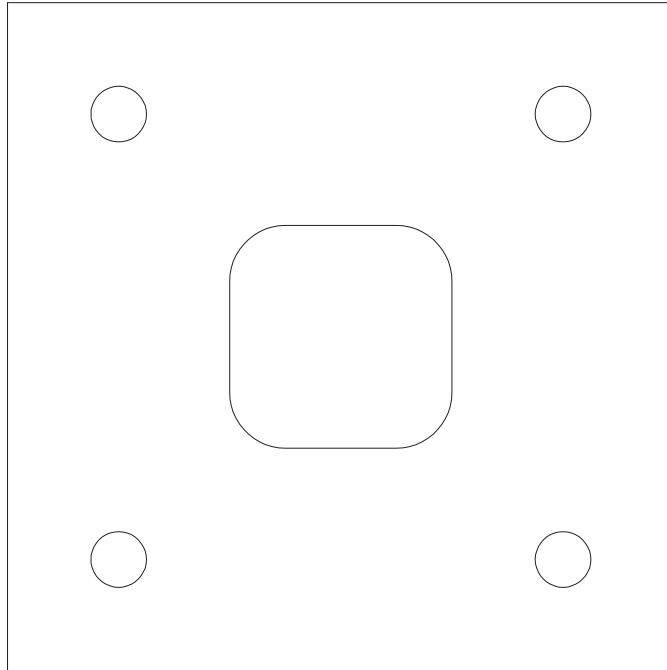
1. Click the **X** button.
2. Notice that AutoCAD displays a dialog box to confirm whether you want to save the drawing — it's your last chance! Move the cursor to choose the **No** button to exit without saving your work. AutoCAD does not record your work to disk, and exits to the Windows desktop.



2D TUTORIAL

With the basics of navigating AutoCAD under your belt, you can now practice drawing in AutoCAD. In the following tutorial, you draw a two-dimensional object.

The figure below illustrates the drawing you construct. It is the plan view of a base with four holes. In the center is a shaft with rounded edges. (In the tutorial following, you create the drawing in 3D.)



The tutorial describes the commands that you enter to create the drawing, which are shown by **boldface** text. (Responses can be in either UPPERCASE or lowercase.) Items enclosed in parentheses (*such as these*) are instructions and are not typed. If ENTER is shown, press the ENTER key on the keyboard. For example:

Command: **line** (Press ENTER.)

Type the **LINE** command, and then press ENTER.

If you make a mistake, click the  **Undo** button on the Quick Access toolbar. You may use it several times to undo each step in reverse order.

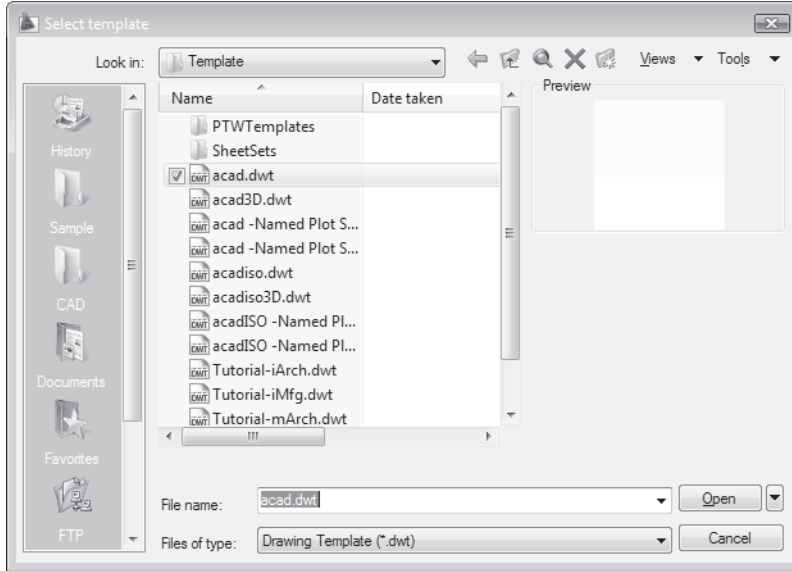
Before you enter a command, the prompt on the command line must be “empty,” with nothing following the colon:

Command:

If not, press ESC. This cancels the current command, and makes AutoCAD ready for your command.

SETTING UP NEW DRAWINGS

1. Start a new drawing by clicking the  **New** button on the Quick Access toolbar. Notice that AutoCAD opens the Select Template dialog box.



2. Choose *acad.dwt*, and then click **Open**.

To draw accurately, you use drawing aids like grid and snap. The *grid* is a visual guide that shows distances. It is like graph paper that uses dots instead of lines. When the grid spacing is set to 1 unit, for example, the drawing area is covered with dots that are spaced one unit apart. The grid is only displayed on the screen; it is not printed.

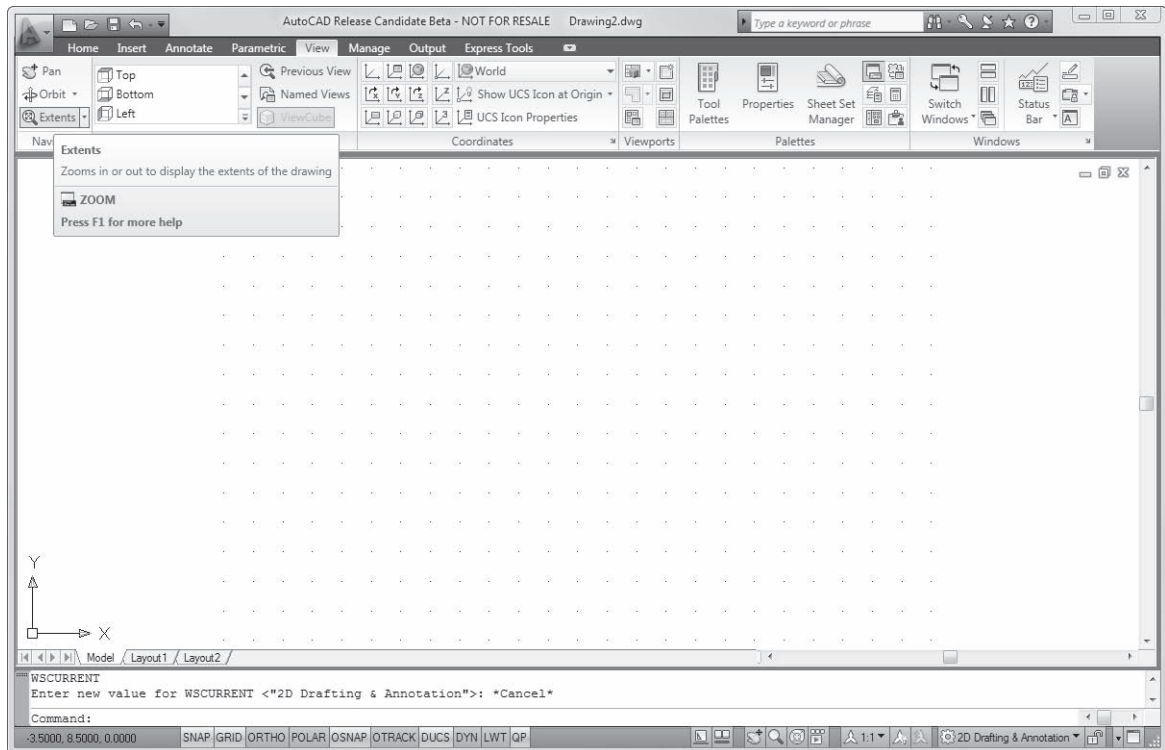
The *snap* is similar to drawing resolution. When on, snap mode causes the cursor to move in precise increments. For example, when the snap is set to 5 units, the cursor moves in a five-unit increment across the drawing. (The grid and snap can have either the same or different spacing from each other.)

You turn on these aids through the status bar, as follows:



3. On the status bar, click the  **Snap** button to turn it on. When status bar buttons are on, they look blue.
4. Again on the status bar, click the  **Grid** button to turn it on. Notice that a grid of fine dots fills the drawing area.

The default spacing for snap and grid is 0.5 units. You can change these values for other drawings through the Drafting Settings dialog box by entering the DSETTINGS command.



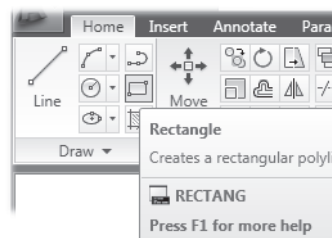
5. If the dots don't fill the drawing area, click the  Zoom Extents button, found in the View tab's Navigate panel.
6. Move the mouse to see the crosshairs cursor move in increments of half units.

DRAWING THE BASEPLATE

The baseplate is a square. You can draw it with the **RECTANGLE** command, which is AutoCAD's command for drawing rectangles and squares.

AutoCAD needs to know the location of the square's two opposite corners, as follows:

1. From the Home tab's Draw panel, click the **Rectangle** button.

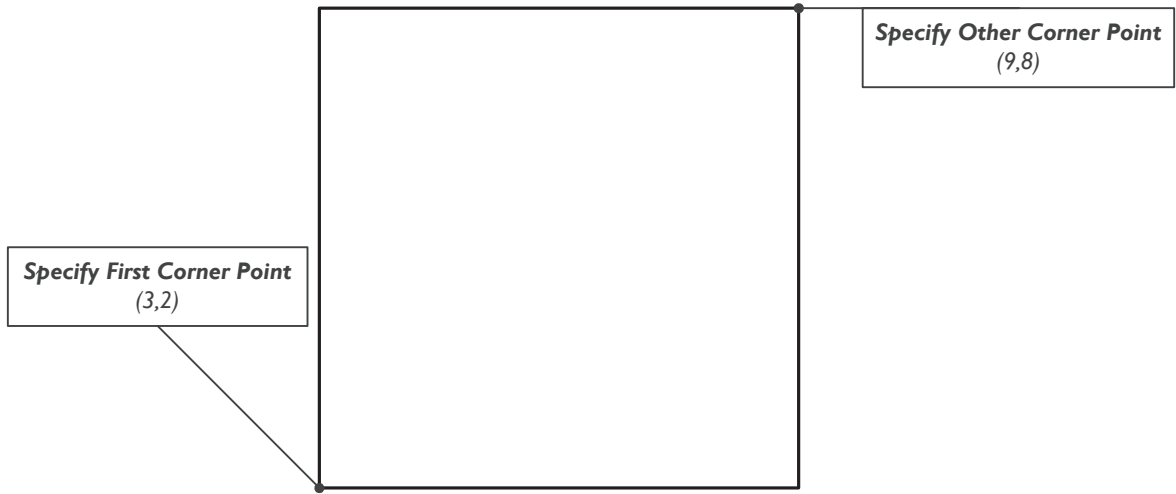


As an alternative, enter the **RECTANGLE** command at the keyboard:

Command: **rectangle** (Press ENTER.)

2. AutoCAD needs to know where you want the square placed. The first corner uses the x,y coordinates of 3,2:
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: **3,2** (Press ENTER.)

3. The opposite corner is located at x,y coordinates of 9,8:
Specify other corner point or [Dimensions]: **9,8** (Press ENTER.)
Your drawing should look similar to the figure below.



Unlike the **LINE** command, the **RECTANGLE** command stops by itself; there is no need to press **ESC**.

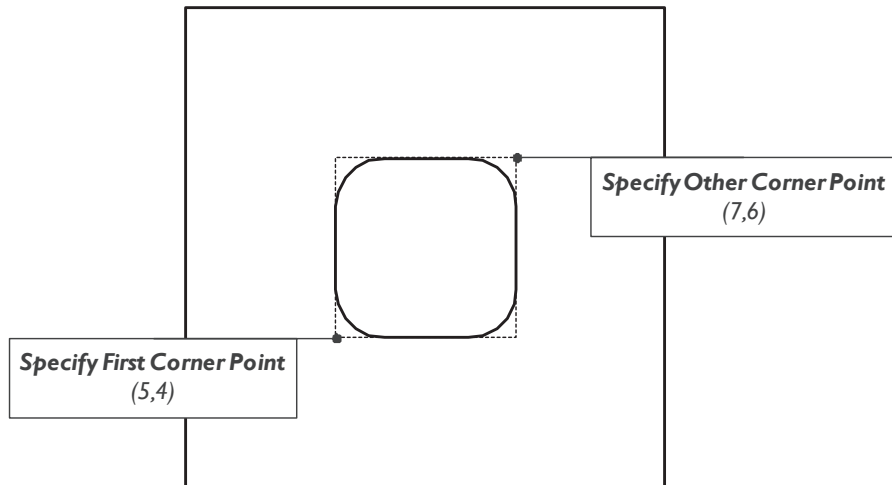
DRAWING THE SHAFT

The shaft is also a square, but with rounded corners. The **RECTANGLE** command can draw squares with squared-off corners as above, rounded off, or cut off.

Repeat the **RECTANGLE** command to draw the square shaft. In addition to specifying the two corners, you need to tell AutoCAD the radius of the rounded corners.

1. Press the **SPACEBAR**. This repeats the previous command — a handy shortcut that saves you typing the name a second time.
Command: (Press **SPACEBAR**.)
RECTANGLE
2. Rounded corners are known in drafting as “fillets,” and the size is specified by a radius. To specify the radius, enter the **Fillet** option:
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: **fillet** (Press ENTER.)
3. These fillets are 0.5 inches in radius:
Specify fillet radius for rectangles <0.0000>: **.5** (Press ENTER.)
4. With the fillet radius set, you can now specify the corners of the square:
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: **5,4** (Press ENTER.)
Specify other corner point or [Dimensions]: **7,6** (Press ENTER.)

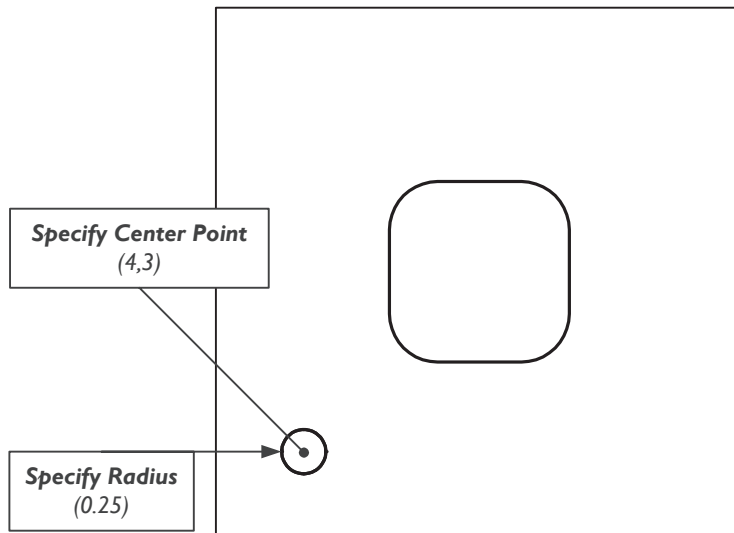
Your drawing should look similar to the figure below.



DRAWING HOLES

Now draw the “holes” in the baseplate. Holes are drawn with circles.

To draw circles, AutoCAD needs to know two things from you: (1) the position in the drawing, based on the center point; and (2) the radius of the circle.



1. From the Home tab's Draw panel, click the  **Circle** button.
2. The first circle is located at 4,3 in the drawing, and so you specify its x,y coordinates like this:
Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **4,3** (Press ENTER.)
3. The circle has a radius of 0.25 units:
Specify radius of circle or [Diameter]: **0.25** (Press ENTER.)

COPYING CIRCLES

To draw the other three circles, you don't repeat the **CIRCLE** command; instead, it is faster to make copies of the first one with the **COPY** command.

AutoCAD needs to know two things from you for making copies: (1) which object(s) to copy, and (2) the location(s) to place copies. Often, the location is specified as the distance between the original and the copied object.

1. Select the object to be copied using AutoCAD's **COPY** command together with an object selection method called "Last" (or "L" for short). It selects the last-drawn object visible on the screen, as follows:

Command: **copy** (Press **ENTER**.)

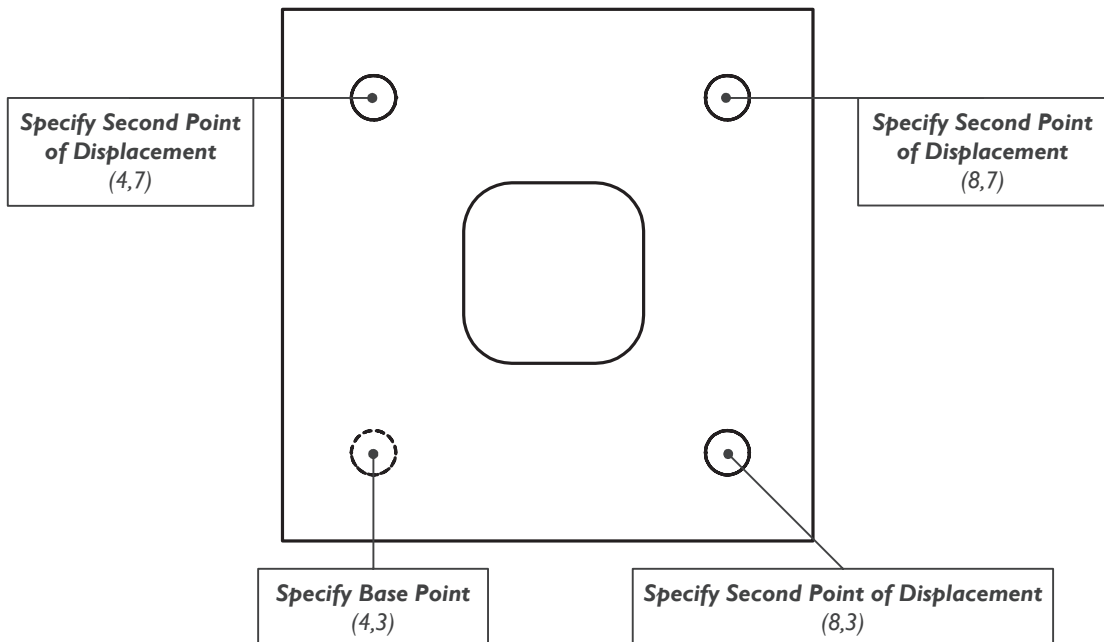
Select objects: **L** (Press **ENTER**.)

Notice that the circle changes its look, and is now made of dashed lines.



This effect is called "highlighting," and is how AutoCAD indicates selected objects.

2. Press **ENTER** to tell AutoCAD you have finished selecting objects.
1 found Select objects: (Press **ENTER** to end object selection.)
3. To specify the distance that the circle should be copied, you tell AutoCAD the *base point* and a *displacement*. The base point is the point from which the copying takes place:
Specify base point or [Displacement] <Displacement>: **4,3** (Press **ENTER**.)



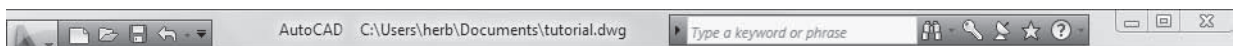
4. The displacement is the point at which the copy is placed:
Specify second point of displacement or <use first point as displacement>: **8,3** (Press **ENTER**.)
5. The **COPY** command repeats its displacement prompt so that you can make multiple copies:
Specify second point of displacement....: **8,7** (Press **ENTER**.)
Specify second point of displacement....: **4,7** (Press **ENTER**.)

6. Press **ESC** to tell AutoCAD you have finished making copies:
Specify second point of displacement...: (Press **ESC** to end the command.)
Your drawing should look similar to the figure illustrated earlier.

SAVING THE DRAWING

With the drawing complete, it is time to save your valuable work.

1. On the Quick Access toolbar, click the  **Save** button.
2. In the Save Drawing As dialog box, enter “tutorial” for the **File name**.
3. Click **Save** to save the drawing.
On the title bar, notice that the name changes from the generic *drawing1.dwg* to *tutorial.dwg*, perhaps prefixed by the path name.

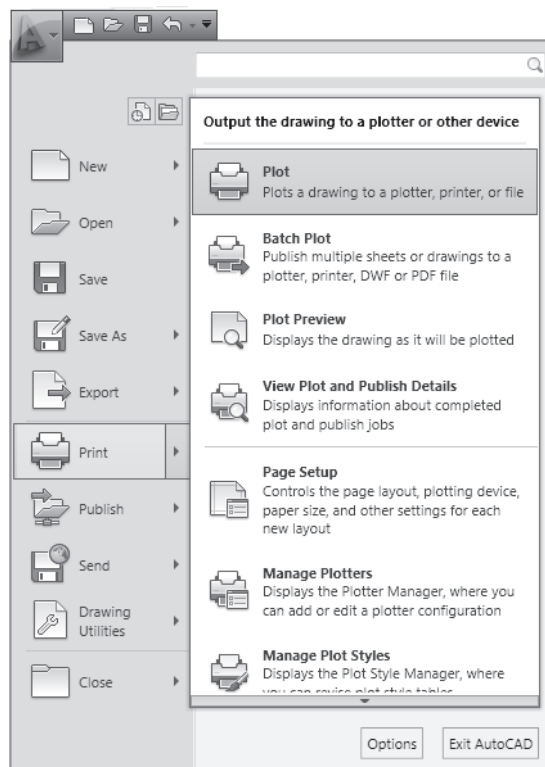


PLOTTING THE DRAWING

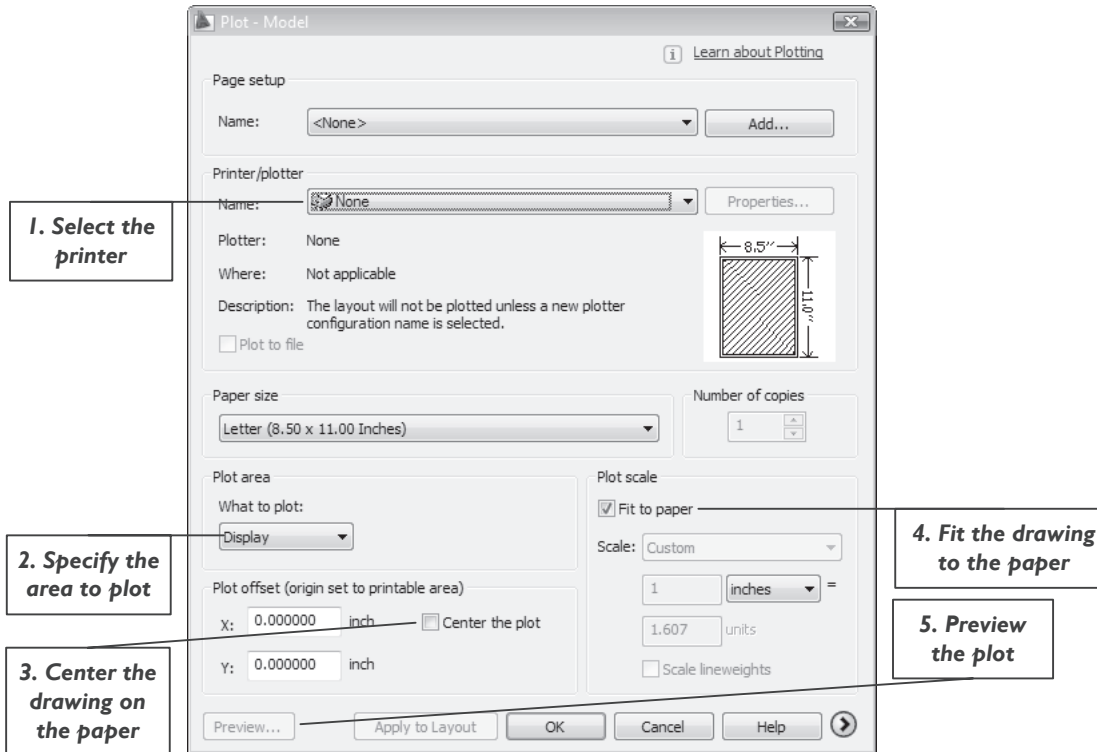
With the drawing saved, print out a copy to show your fine work to your instructor, family, and friends! This is done with the **PLOT** command.

Before printing, AutoCAD needs to know: (1) the printer on which the drawing will be plotted, (2) the view to plot, and (3) the size. There are many other options, but these three are the most important.

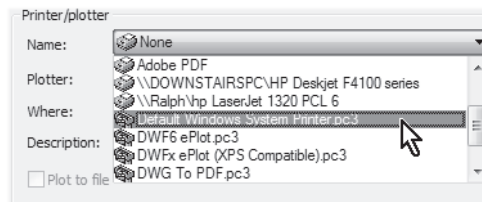
1. Click the big red **A**, and then choose **Print**.



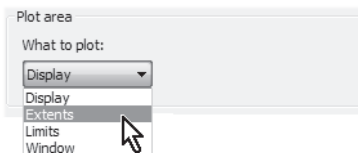
- From the options lists, choose **Plot**. AutoCAD displays the Plot dialog box with its many options. It can be a little overwhelming, so I've highlighted the five areas you'll need for this tutorial.



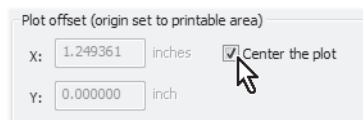
- In the Printer/plotter area, select a printer from the **Name** droplist. "Default Windows System printer.pc3" is usually the safe choice. This plots the drawing on the same printer your computer uses for all other printing tasks.



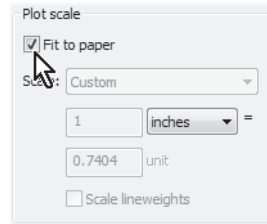
- In the Plot Area section, select **Extents** from the What to Plot droplist. This option ensures the entire drawing is plotted.



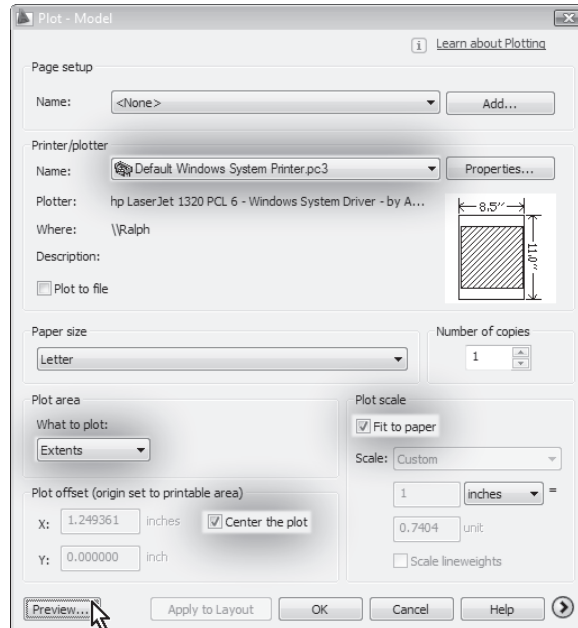
- In the Plot Offset section, select **Center the plot** to center the drawing on the paper.



- In the Plot Scale area, select **Fit to Paper**. This ensures the drawing fits the paper, no matter the size of paper or drawing.

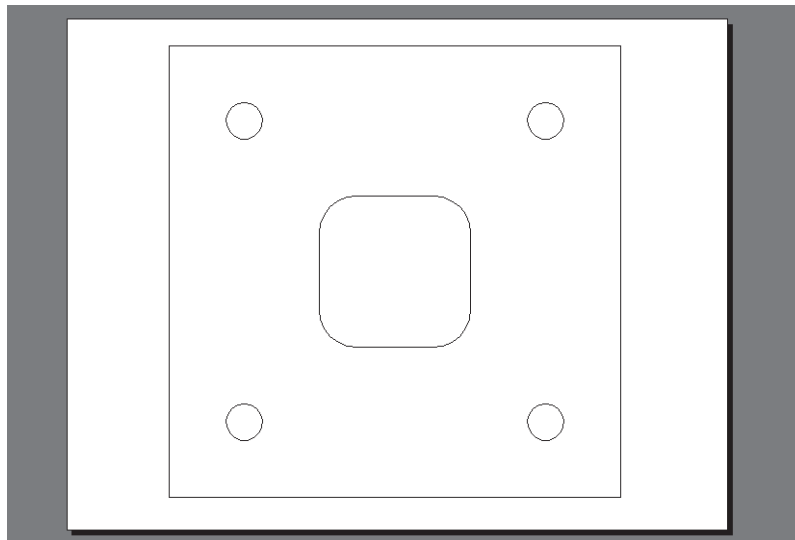


- At this point, the dialog box's options should look like those illustrated below.



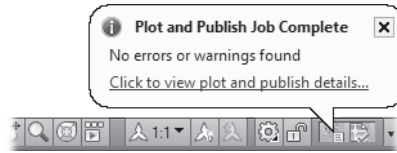
To ensure the plot will work out correctly, click the **Preview** button. This lets you check for errors before committing (or wasting) paper.

AutoCAD shows you what the plot will look like, as illustrated by the figure below.



- Press **ESC** to exit the print preview mode.
Press **ESC** or **ENTER** to exit, or right-click to display shortcut menu. (Press **esc**.)

9. Back in the Plot dialog box, choose **OK** to start the plot. AutoCAD displays a dialog box showing the progress of the plot. When done, a yellow alert balloon appears at the right end of the status bar, reporting on the success (or failure) of the plot.

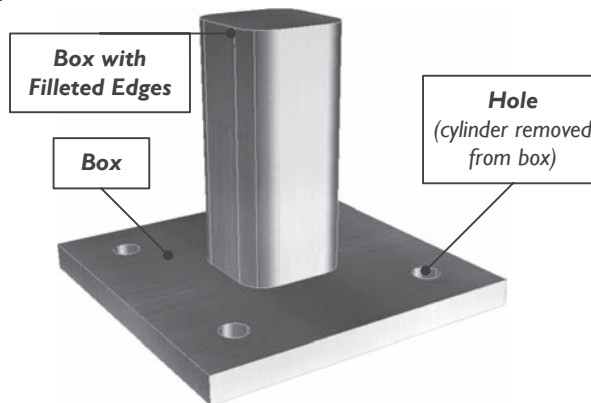


After a moment, the drawing should emerge from your printer.

10. Save the drawing with the **QSAVE** command, and then exit AutoCAD, as you learned earlier in this chapter.

3D TUTORIAL

Let's repeat the tutorial to draw the same part but in three dimensions. The figure below illustrates the result: the baseplate with its four holes and the shaft with rounded edges.

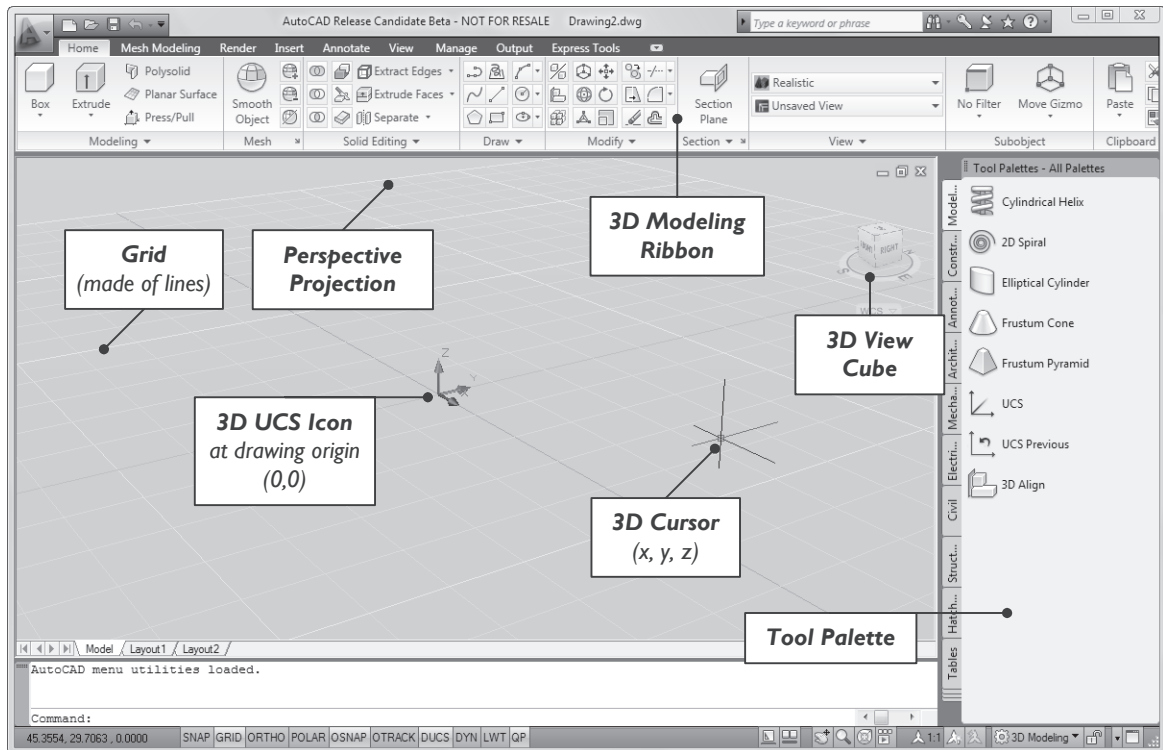


Modeling objects in 3D is different from drafting in 2D, because 3D objects are made of *solid primitives*, which are elements like boxes and cylinders. For instance, the base is made of a box, and the shaft is a second box with four of its edges filleted. Holes are made of cylinders that are then removed from the base.

Begin your work in the 3D modeling workspace.

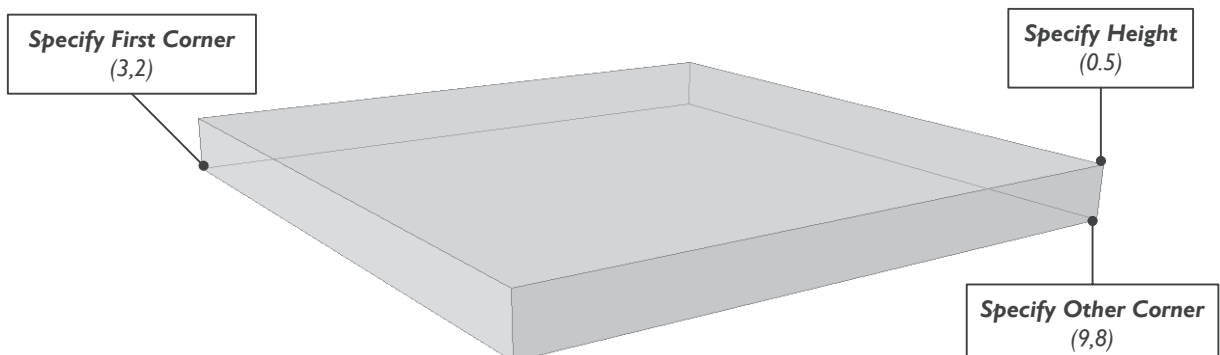
1. From the Workspaces button on the status bar, choose **3D Modeling**.
2. Click the  **New** button on the Quick Start toolbar, and then select the *acad3d.dwt* template file.

Notice that AutoCAD's environment changes significantly.



Ensure dynamic input is turned off; otherwise your results may differ from this tutorial. Click the **DYN** button on the status bar.

3. Enter the **BOX** command at the keyboard:
Command: **box** (Press ENTER.)
4. AutoCAD next needs to know where to place the box. One corner is at the x,y coordinates of 3,2:
Specify first corner or [Center]: **3,2** (Press ENTER.)
5. The opposite corner is located at x,y coordinates of 9,8:
Specify other corner or [Cube/Length]: **9,8** (Press ENTER.)
6. The height is 0.5:
Specify height or [2Point]: **0.5** (Press ENTER.)



7. Your drawing should look similar to the figure above. If necessary, enter the **ZOOM** command, and select the **All** option:

Command: **zoom** (Press ENTER.)

Enter option [All/Extents/Window/Previous] <real time>: **all** (Press ENTER.)

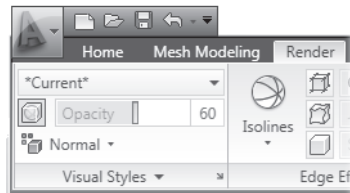
MODELING THE SHAFT

The shaft is also a box, but with rounded edges. Unlike **RECTANGLE**, the **BOX** command doesn't have a built-in filleting option, so you'll use the **FILLET** command later.

1. Repeat the **BOX** command.

Command: **box**

2. To make the boxes easier to see and work with, turn on transparency. In the ribbon's Render tab, click the **Opacity** button (found in the Visual Styles panel).

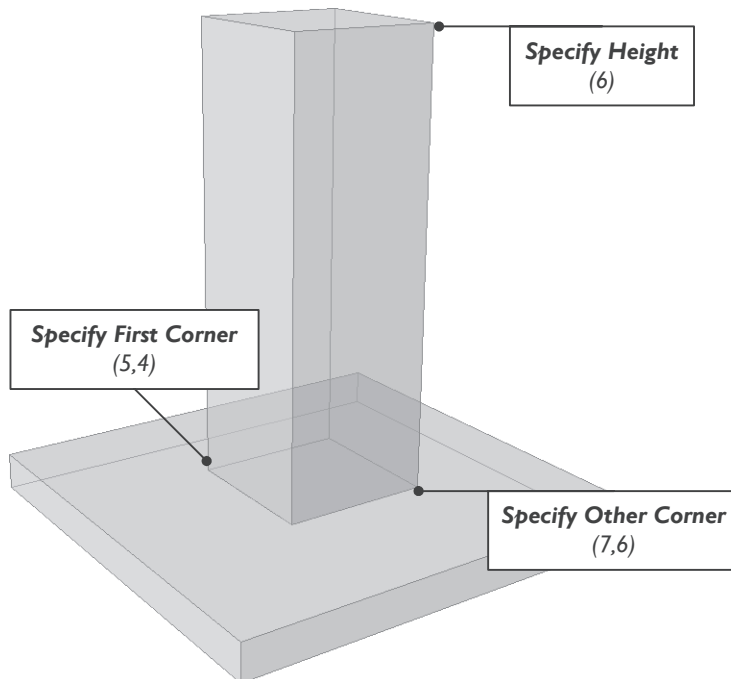


3. Specify the corners and height of the box by typing x,y coordinates, as follows:

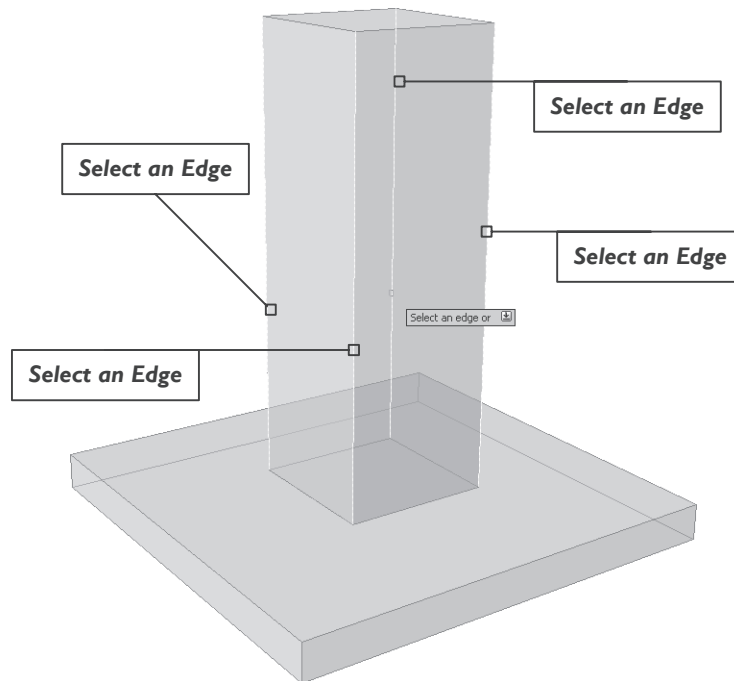
Specify first corner or [Center]: **5,4** (Press ENTER.)

Specify other corner or [Cube/Length]: **7,6** (Press ENTER.)

Specify height or [2Point]: **6** (Press ENTER.)



4. To round the edges of the box, enter the **FILLET** command:
 Command: **fillet**
 Current settings: Mode = TRIM, Radius = 0.0000
5. Enter the “m” option (short for multiple), because you will be filleting multiple (four) edges:
 Select first object or [Undo/Polyline/Radius/Trim/Multiple]: **m**
6. The fillets are 0.5 inches in radius:
 Enter fillet radius <0.0000>: **0.5** (Press ENTER.)
 Notice that the cursor changes to a small square; this is called the “pick” cursor.
7. Now tell AutoCAD which edges to fillet by picking each one: you must position the square pick cursor over each edge, and then click the left mouse button:
 Select an edge or [Chain/Radius]: (Pick an edge.)



As you do, the edge turns white. Repeat for the other three edges:

Select an edge or [Chain/Radius]: (Pick another edge.)

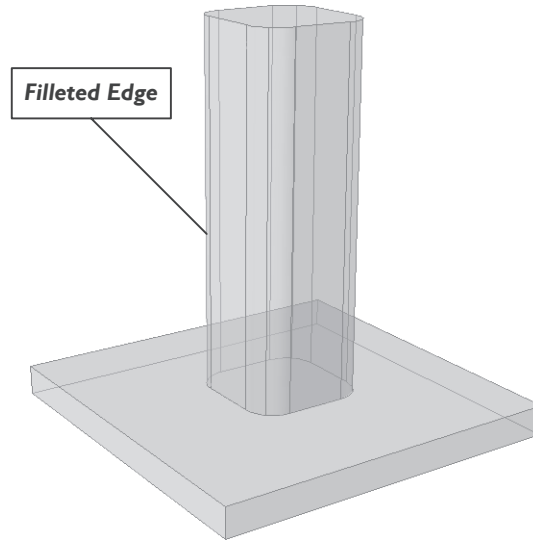
Select an edge or [Chain/Radius]: (Pick a third edge.)

Select an edge or [Chain/Radius]: (Pick the last edge.)

8. With the four edges selected, press **ENTER** to exit selection mode:
 Select an edge or [Chain/Radius]: (Press ENTER to exit selection mode.)
 4 edge(s) selected for fillet.

9. Press **ENTER** a second time to exit the command.

Select first object or [Undo/Polyline/Radius/Trim/Multiple]: (Press **ENTER** to exit the command.)



Notice that the column has rounded edges.

MODELING HOLES

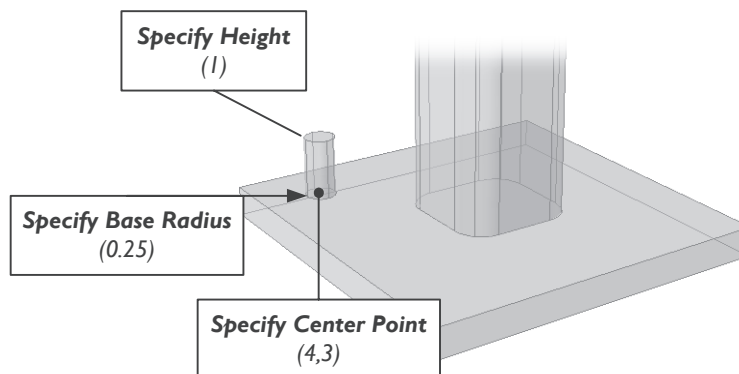
The next step is to draw the holes in the baseplate. In AutoCAD, holes are constructed by drawing, then removing, cylinder shapes.

1. Enter the **CYLINDER** command:

Command: **cylinder**

2. The cylinder is located at 4,3 in the drawing:

Specify center point of base or [3P/2P/Ttr/Elliptical]: **4,3** (Press **ENTER**.)



3. The circle has a radius of 0.25 units:

Specify base radius or [Diameter]: **.25** (Press **ENTER**.)

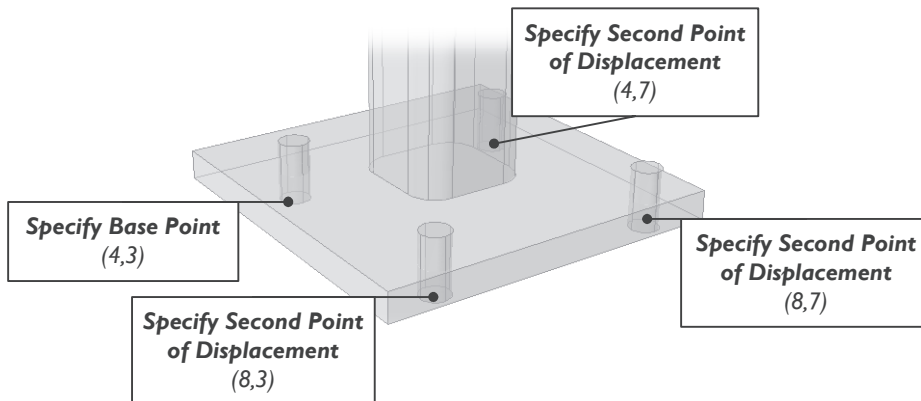
4. The height does not matter, just as long as it is taller than the base:

Specify height or [2Point/Axis endpoint] <6.0>: **1** (Press **ENTER**.)

COPYING CYLINDERS

To draw the other three cylinders, use the **COPY** command to copy the first cylinder to the other locations on the baseplate.

1. Start the **COPY** command, and then select the last-drawn cylinder with the **L** object selection option, as follows:
 Command: **copy** (Press **ENTER**.)
 Select objects: **L** (Press **ENTER**.)
 1 found Select objects: (Press **ENTER** to end object selection.)
2. Recall that the *base point* is the point from which the copying takes place:
 Specify base point or [Displacement] <Displacement>: **4,3** (Press **ENTER**.)



3. The *displacement* is the point at which each copied cylinder is placed:
 Specify second point of displacement or <use first point as displacement>: **8,3** (Press **ENTER**.)
 Specify second point of displacement....: **8,7** (Press **ENTER**.)
 Specify second point of displacement....: **4,7** (Press **ENTER**.)
 Specify second point of displacement....: (Press **ESC** to end the command.)

SUBTRACTING CYLINDERS TO CREATE HOLES

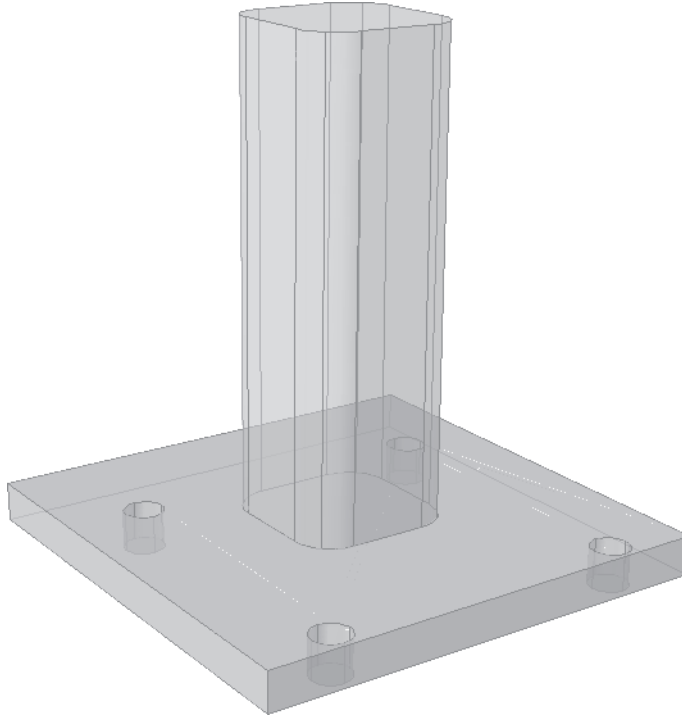
With the four cylinders in place, you now use the **SUBTRACT** command to turn them into holes.

1. Enter the **SUBTRACT** command:
 Command: **subtract**
 AutoCAD first asks you for the object(s) to subtract *from*; that would be the baseplate:
 Select solids and regions to subtract from ..
 Select objects: (Select the baseplate.)
 Select objects: (Press **ENTER** to end object selection.)
2. Now AutoCAD needs to know which objects to subtract; that would be the four cylinders:
 Select solids and regions to subtract ..
 Select objects: (Select one cylinder.)
 Select objects: (Select another cylinder.)
 Select objects: (Select a third cylinder.)

Select objects: *(Select the fourth cylinder.)*

Select objects: *(Press ENTER to end object selection.)*

Notice that AutoCAD removes the cylinders from the baseplate, creating four holes.



3. The final step is to join the shaft with the baseplate. This is done with the **UNION** command.

Command: **union**

Select objects: *(Select the base plate.)*

Select objects: *(Select the shaft.)*

Select objects: *(Press ENTER to end object selection.)*

The model does not look any different, but it is now a single object.

4. Save the drawing as “3d Tutorial,” and then print it, if you wish.

So, that was a quick start to using AutoCAD.

Welcome to the rest of *Using AutoCAD 2010*. The remaining chapters take you step by step through the program. Before long, you’ll be using AutoCAD to create drawings like a pro.

Enjoy!

EXERCISES

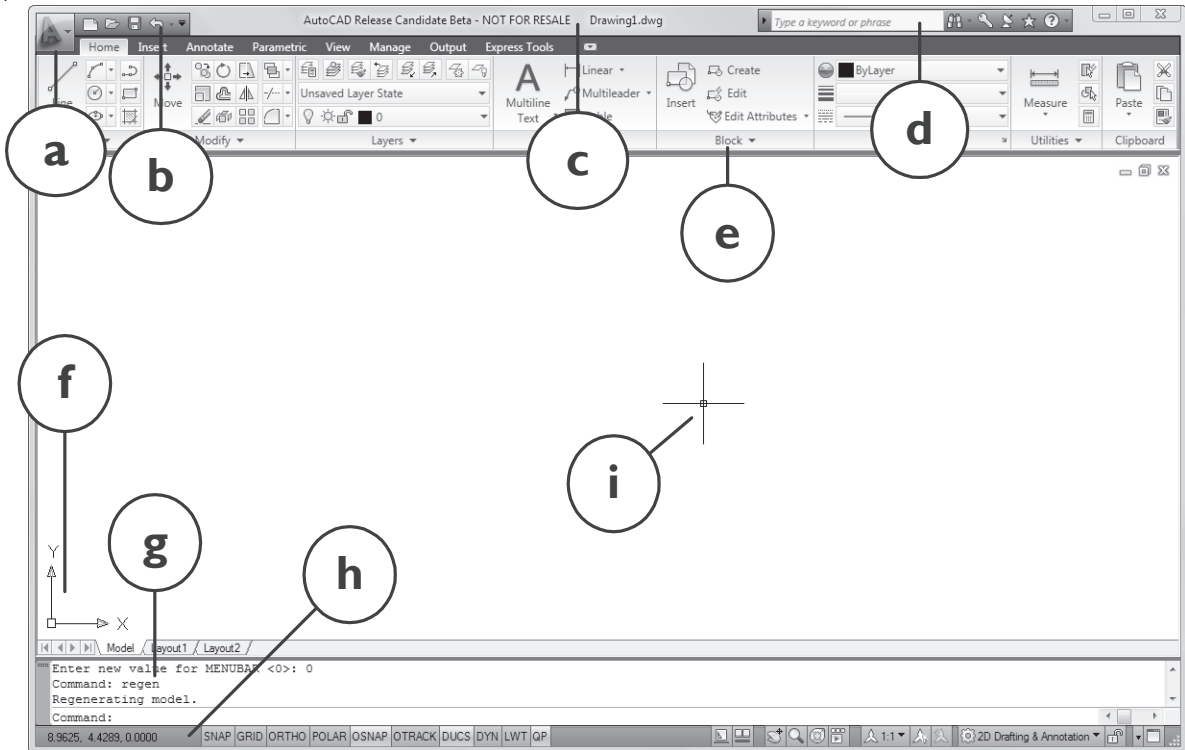
1. For this exercise, start AutoCAD.
(If the Start New Drawing dialog box appears, click **Cancel**.)
With the **LINE** command, draw several shapes:
 - a. Rectangle.
 - b. Triangle.
 - c. Irregular polygon (any shape you like).
2. Use the **U** command. What happens to the last object you drew?
3. Press function key **F9**, and then look at the status line.
Does the button look blue (turned on)?
4. Repeat the **LINE** command, and again try drawing these shapes:
 - a. Rectangle.
 - b. Triangle.
 - c. Irregular polygon.Do you find it easier?
5. Draw a rectangle with the **RECTANGLE** command using these parameters:
Corner **2,3**
Other corner **6,7**
6. Draw a circle with the **CIRCLE** command using these parameters:
Center point **4,5**
Radius **0.75**
Is the circle drawn “inside” the box?
7. Print your drawing with the **PLOT** command.
Does the plot look like the drawing on your computer screen?

CHAPTER REVIEW

1. What are AutoCAD drawings constructed with?
2. Name three ways in which commands can be entered:
 - a.
 - b.
 - c.
3. Describe the purpose of the 'Command:' prompt.
4. How do you exit from print preview mode?
5. What is the purpose of the **LINE** command?
6. How is the **U** command helpful?
7. Describe how to cancel commands.
8. What does the mouse control?
9. What is an *icon*?
10. Name two methods by which to determine the function of ribbon buttons:
 - a.
 - b.
11. What are *tooltips*?
12. What are *flyouts*?
13. Can toolbars be moved around the AutoCAD window?
14. Which commands close drawings without exiting AutoCAD?
15. Which commands exit AutoCAD?
16. Which command saves drawings?
17. Which command starts new drawings?
18. Describe how snap and grid are useful:
 - Snap
 - Grid
19. Is the grid plotted?
20. List three things AutoCAD needs to know, as a minimum, before plotting drawings:
 - a.
 - b.
 - c.
21. Explain the advantage of the **PLOT** command's **Fit to Scale** option.
22. Why is the **Preview** option environmentally friendly?
23. How do you exit from print preview mode?

24. Identify the user interface elements illustrated below:

- a.
- b.
- c.
- d.
- e.
- f.
- g.
- h.
- i.



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Understanding CAD Concepts

CAD software contains concepts that distinguish it from other software programs. In part, the uniqueness stems from CAD being based on *vectors*, geometric objects that have length and direction. Using AutoCAD means knowing how to manipulate vectors; before you can manipulate vectors successfully, you need to understand how AutoCAD operates.

This chapter covers the following activities:

- Touring the AutoCAD user interface.
- Saving and restoring user interface configurations.
- Understanding the information AutoCAD presents to you.
- Entering commands.
- Learning shortcut keystrokes and command aliases.
- Picking points on the screen.
- Using mouse buttons and menus.
- Identifying the elements of dialog boxes.
- Understanding the draft-edit-plot cycle.
- Recognizing AutoCAD's coordinate systems.
- Entering coordinates from the keyboard and on the screen.

NEW TO AUTOCAD 2010 IN THIS CHAPTER

- The ribbon and other aspects of its user interface have changed.
- A new application menu replaces the menu browser.
- There are new commands and aliases.

AUTOCAD USER INTERFACE

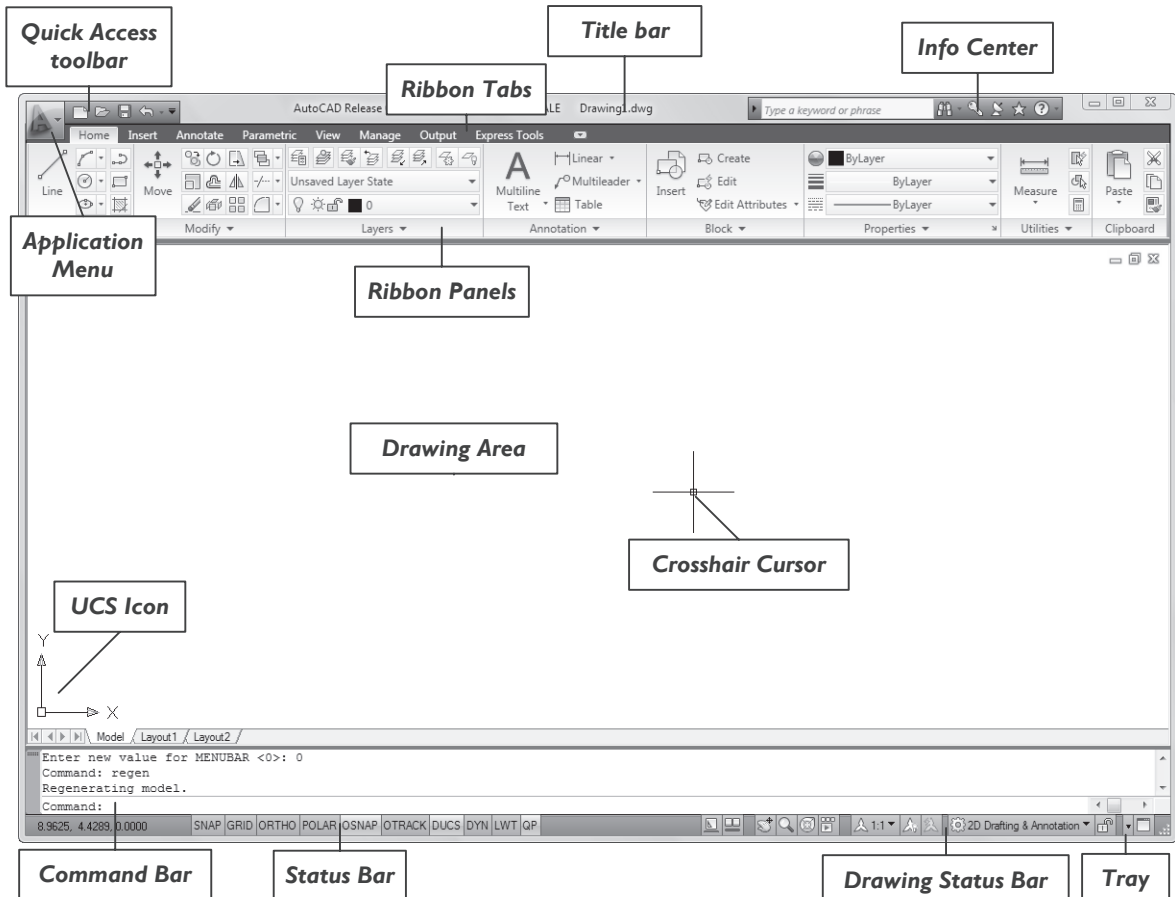
Your drawing sessions begin by starting the AutoCAD program. As described in Chapter 1, “Quick Start in AutoCAD,” double click the AutoCAD icon to start the program.



(If there is no icon on the Windows desktop, click the taskbar’s **Start** button, and then choose **Programs**. Choose the **Autodesk** folder, the **AutoCAD 2010** folder, and then the **AutoCAD 2010** program.)

When AutoCAD loads, you see either a blank drawing or one or more dialog boxes — depending on how it was configured. (If dialog boxes appear, close them.) You see the *drawing area*, the window in which you draw.

AutoCAD will look different, depending on which workspace is loaded. The “2D Drafting & Annotation” workspace looks like this:



TOURING THE USER INTERFACE

AutoCAD displays information in several areas. We looked at some of them in the previous chapter; now let’s look at the remaining areas of interest, starting at the top and then moving down the window.

TITLE BAR

At the very top of the AutoCAD window is the *title bar*. The center of the title bar tells you the name of the software, AutoCAD 2010, as well as the file name of the current drawing, such as “Drawing1.dwg.”



The large red A in the upper left corner hides the application menu described in detail in Chapter 1. Next to it is the Quick Access toolbar, also described in Chapter 1.

Info Center

Near the right side of the title bar is the Info Center. (It replaces the Info palette from earlier releases of AutoCAD.) When you enter terms, it accesses the online help.



From left to right, the buttons perform the following functions:

Button	Meaning
	Collapses the InfoCenter.
	Searches the help files, as well as resources on Autodesk’s Web site.
	Displays a list of search options.
	Accesses Autodesk’s subscription center by Web browser.
	Lists information from Autodesk’s blogs and online learning.
	Collects “favorites,” often used help resources.
	Displays online help (<i>new to AutoCAD 2010</i>).

Window Controls

At the far right are three window controls that are common to every Windows application. (A similar set of buttons is available for each drawing window.) From left to right, the buttons perform the following functions:



Button	Meaning	Related Commands
	Minimizes the AutoCAD or drawing window.	
	Restores the AutoCAD or drawing window.	
	Closes the AutoCAD window, and closes the drawing window.	EXIT command. CLOSE command.

Below the title bar is the ribbon, which replaces the menu bar and toolbars from earlier releases of AutoCAD. You can toggle the display of the ribbon with the **RIBBON** and **RIBBONCLOSE** commands. It also replaces the Dashboard of AutoCAD 2007 and 2008. (Indeed, the **DASHBOARD** and **DASHBOARDCLOSE** commands now toggle the display of the ribbon.) You can simulate the Dashboard by undocking the ribbon.

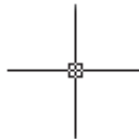
You can bring back the menu bar by setting the new **MENUBAR** system variable to 1. It appears in its traditional location below the title bar. Toolbars can be returned with the **-TOOLBAR** command.

DRAWING AREA

The drawing area is where you draw, but it also displays information.

Cursors

The *crosshair cursor* is most important element of the drawing area, because it shows you where you are in the drawing. The large crosshair lines help you easily locate the cursor.



The small square at the cursor’s center indicates that you can *select* (or “pick”) objects in the drawing for editing. (Pressing the left mouse button selects the objects.) This square is called the “pickbox.” Its size can be varied through a system variable named **PICKBOX**.

When you move the crosshair cursor out of the drawing area, it changes to an *arrow cursor*, with which you are probably familiar from other Windows software.



The arrow cursor lets you select items outside of the drawing area, such as buttons in the ribbon and options in dialog boxes.

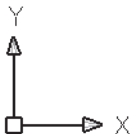
Cursor Icons

From time to time, AutoCAD displays small icons near the crosshair cursor — or in place of it. You can expect to see these cursor icons, plus many more:

Icon	Meaning	Related Commands
	Specifies orthographic drawing mode.	ORTHO
	Specifies that object or layer is locked.	LAYER
	Specifies that table is linked to external data file.	DATALINK
	Specifies that drawing is being panned.	PAN
	Specifies that drawing has panned to its extents.	PAN
	Specifies that drawing is in realtime zoom.	ZOOM
	Specifies that object has annotative scale factor.	ANNOSCALE
	Specifies that object has two or more annotative scales.	

UCS Icon

The lower-left corner sports an L-shaped arrow. It is called the *UCS icon*; UCS is short for “user coordinate system.”



The UCS icon is useful primarily for drawing in three dimensions; it helps to orient you in three-dimensional space.

The icon is not needed for 2D drafting, and you can turn it off with the **UCSICON** command. Type the command name, and then enter the **Off** option, as follows:

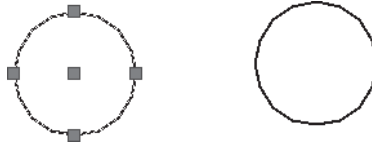
```
Command: ucsicon (Press ENTER.)
Enter an option [ON/OFF/All/Noorigin/ORigin/Properties] <ON>: off (Press ENTER.)
```

Other Screen Markings

In addition to the cursor and icons, the drawing area often displays additional markings.

Highlights and Grips

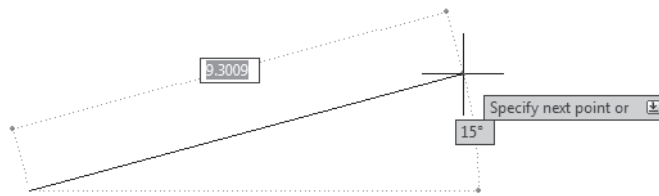
When you select objects in drawings, AutoCAD highlights them with a dotted line. The highlighting is feedback that confirms which object(s) were selected. The figure below shows objects that are selected (highlighted, and with grips) and not selected (normal).



Most times, selected objects also show *grips*. These small squares allow you to edit objects directly without commands. You can move or copy objects using grips, as well as stretch, mirror, and rotate them, and perform other operations. Grips editing is detailed in a later chapter.

Dynamic Input

AutoCAD can display prompts at the cursor. This is called “dynamic input,” and was detailed in Chapter 1.



Dynamic prompts are turned on and off through the  (or DYN) button on the status bar.

Grid

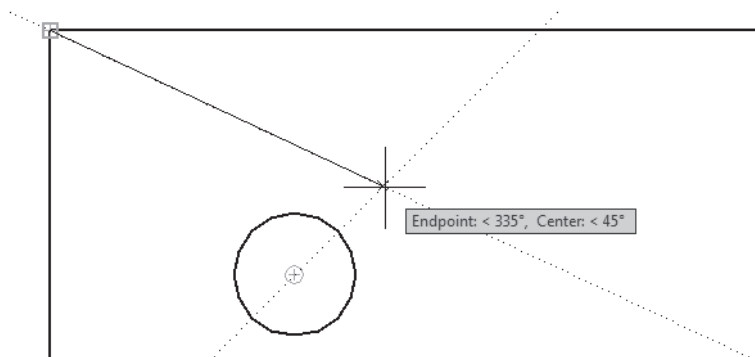
The grid is a visual aid that helps you gauge distances in drawings. (It was described in Chapter 1.) It is turned on and off through the  (or GRID) button on the status bar.

In 2D drawings, the grid consists of an array of evenly spaced dots. In 3D drawings, the grid looks like graph paper, consisting of spaced lines.

Tracking Lines

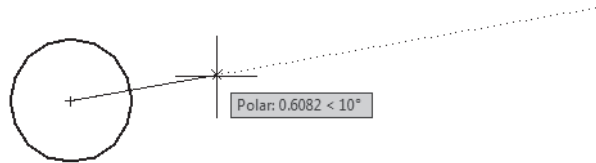
Tracking lines are dotted lines that radiate from the cursor. These show you geometric relationships between objects. There are two types of tracking: object snap and polar.

Object snap tracking shows you relationships between the cursor and geometric features of objects, such as their endpoints, centers, and intersections. In the figure below, the dotted lines track relationships between the cursor's current location and the center and endpoint of nearby objects. The accompanying tooltip report the angles to the “Endpoint” of a line and the “Center” of the circle.



Object snap tracking is turned on with the  (or **OTRACK**) button.

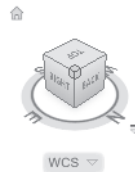
Polar tracking lets you know when the cursor is at discrete angles from the horizontal, such as at 15 or 90 degrees. In the figure below, the dotted line indicates tracking at 10 degrees; the tooltip reporting the polar distance and angle.



Polar tracking is turned on with the  (or **POLAR**) button on the status bar.

ViewCube

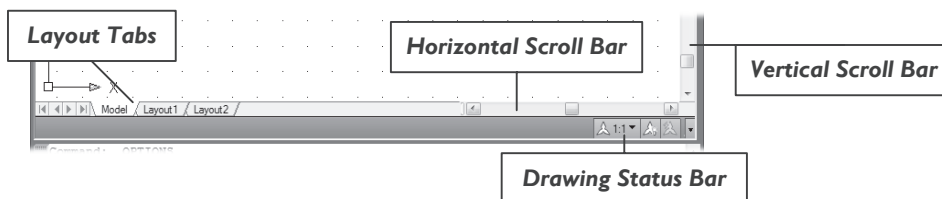
The viewcube quickly changes viewpoints in 3D drawings, and so is displayed only when you are working in a 3D environment.



When you manipulate the viewcube by clicking on its faces or corners, AutoCAD immediately changes the drawing's viewpoint to match.

LAYOUT TABS AND SCROLL BARS

Below the drawing area are three or more tabs labeled Model, Layout1, and so on. (In AutoCAD 2009, they were turned off by default.) These tabs let you switch quickly between model space and paper space. You learn more about layouts later in this book.



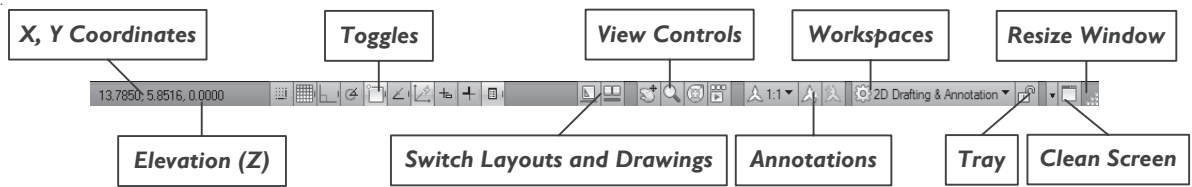
Scroll bars let you move the drawing around the screen — called “panning.” You can click or drag the bars to pan the drawing horizontally and vertically. Scroll bars are turned off by default; to turn them on, enter the **OPTIONS** command, choose the **Display** tab, and then select the **Display Scroll Bars in Drawing Window** option.

Sometimes, the drawing status bar appears below the scroll bar. (If not, then it is located at the right end of the status bar.) This row of icons is related to annotation scaling, described later in this book.

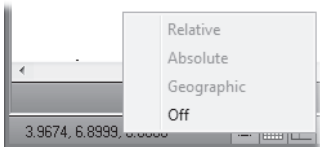
Below the layout tabs is the *command prompt*. This is the traditional area for entering commands, and one place where AutoCAD responds with prompts and messages, as described in Chapter 1.

STATUS BAR

At the bottom of the AutoCAD window is the *status bar*. This bar reports the coordinates of the cursor location and the status of the drawing through buttons.



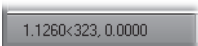
Many of these settings are turned on and off simply by clicking the buttons. Additional controls are available by right-clicking buttons and selecting options from shortcut menus. For example, to change the way coordinates are displayed, right-click them and choose another format.



X, Y Coordinates and Elevation

At the left of the status bar is a set of three numbers. They represent the *coordinates* of the crosshair cursor: the x, y coordinates plus the current elevation (not the z coordinate). The format of the numbers and angles is controlled by the UNITS command, as you will learn in Chapter 3, “Setting Up Drawings.”

The x,y coordinates constantly update as you move the cursor. You can change the display to *relative* (distance<angle) readout through the shortcut menu described above.



If the coordinates look gray and do not update as you move the cursor, then they are turned off; click the coordinate display to turn them on.

Mode Indicators

Next on the status bar are *mode indicators* that look like buttons. These are also known as “toggles,” because they turn modes on and off — like light switches. Clicking a button turns the mode on or off. When on, the mode’s button looks bluish; when off, grayish.

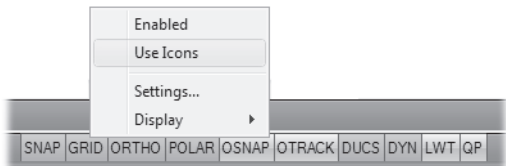


From left to right, the buttons have the following meaning:

Icon	Text	Shortcut	Keystroke	Meaning
	SNAP	F9		Cursor snap is on or off.
	GRID	F7		Grid display is on or off.
	ORTHO	F8		Orthogonal mode is on or off.
	POLAR	F10		Polar snap mode is on or off.
	OSNAP	F3		Object snap modes are on or off.
	OTRACK	F11		Object tracking mode is on or off.
	DUCS	F6		Dynamic user-coordinate system is on or off.
	DYN	F12		Dynamic input mode is on or off.
	LWT	...		Lineweights are displayed or not.
	QP	...		Quick Properties palette is on or off.

As an alternative to clicking buttons on the status bar, you can press the shortcut keystrokes listed by the table above.

The buttons display icons (as shown above) or text labels (below). To change them, right-click any button, and then choose **Use Icons** from the shortcut menu.



Layouts

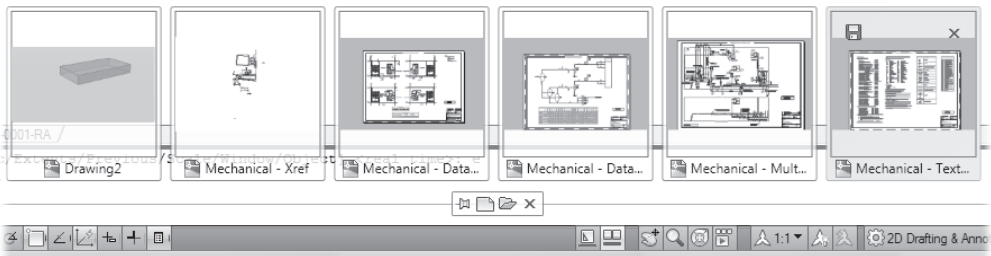
Next to the toggle buttons are two controls that switch visually among drawings and layouts. (Layouts are described later in this book.)



From left to right, the buttons have the following meaning:

Icon	Meaning	Related Command
	Switch visually between drawings.	QVDRAWING
	Switch visually switch between layouts.	QVLAYOUT

Click a button, and AutoCAD displays thumbnail-size previews of other the open drawings or layouts. Click a thumbnail to switch to that drawing.



View Controls

The next four buttons change the view with panning and zooming, as well as access the navigational steering wheel and operate show-motion animations.



From left to right, buttons have the following meaning:

Icon	Meaning	Related Command
	Pans the drawing around the screen.	PAN
	Zooms the drawing larger or smaller.	ZOOM
	Moves around drawings interactively.	NAVSWHEEL
	Creates animated movies with named views.	SHOWMOTION

Annotation Scaling

Most objects in AutoCAD can be drawn and plotted at any size, but some cannot. Text, hatching, and linetypes must look the correct size when plotted. Annotation scaling ensures that these scale-dependent objects display and plot at the correct scale factor.



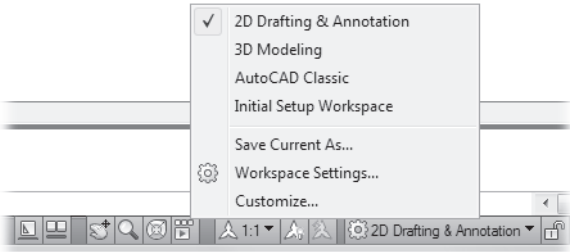
From left to right, the buttons have the following meaning:

Icon	Meaning	Related Command
	Specifies the annotation scale; lists scale factors when clicked.	CANNOSCALE OBJECTSCALE
	Toggles visibility of annotation objects.	ANNOALLVISIBLE
	Adds annoative scale factors automatically.	ANNOAUTOSCALE

Annotative scaling is discussed in detail in Chapter 10, “Placing and Editing Text.”

Workspaces

Workspaces change aspects of the user interface, such as the ribbon, toolbars, and palettes. To change, click the **Workspace** button, and then choose another one. You can create custom workspaces.

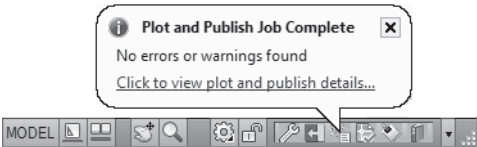


Tray

At the right end of the status bar are icons, many of which appear only when AutoCAD needs to tell you something. For some reason, this part of the status bar gets its own name: the tray. The one icon that is always present is illustrated below — unless turned off via Tray Options.As needed, other icons appear in the tray, as follows:

Icon	Meaning	Related Commands
	Locks palettes and toolbars into position.	LOCKUI
	Indicates that drawing has CAD standards attached.	STANDARDS, CHECKSTANDARDS
	Indicates that tables contain linked attribute data.	DATAEXTRACTION
	Indicates that tables are linked to external data files.	DATALINK
	Indicates that externally-referenced drawings are attached.	XATTACH
	Indicates that drawing is signed digitally.	SIGVALIDATE
	Indicates that drawing is created by Autodesk software.	DWGCHECK
	Indicates that plots are underway or complete.	PLOT, VIEWPLOTDETAILS
	Warns that layers were added from xref drawings.	LAYERNOTIFY
	Indicates that Communications Center is active.	...

Sometimes, icons display yellow alert balloons to warn you of changes and problems.

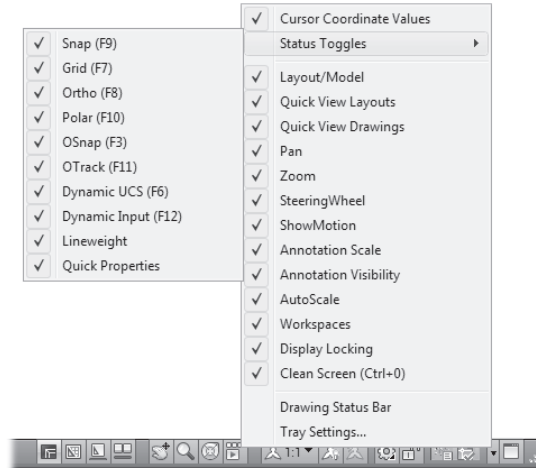


Click the blue underlined text for more information, or click the x in the upper-right corner to dismiss the balloon.

  To keep the toolbars and palettes from shifting on the screen, you can lock them in place. At the far right end of the status bar, click the padlock icon, and then choose which user interface elements to lock: floating and docked toolbars, panels, “windows,” or all. (*Windows* is the old term for palettes, such as the Properties palette — and not the AutoCAD window itself.) Hold down the CTRL key to move locked items temporarily.

Tray Options

Click the arrow button for a menu of options for the status bar and tray.



Cleanscreen & Resize Window

The  **Clean Screen** button maximizes the AutoCAD window, and turns off the title bar and toolbars; click the button a second time to return the window to its former size.

The  **Resize AutoCAD** button changes the size of the AutoCAD window; drag the corner with the mouse; the button is not present when AutoCAD is in full-screen mode.

ENTERING COMMANDS

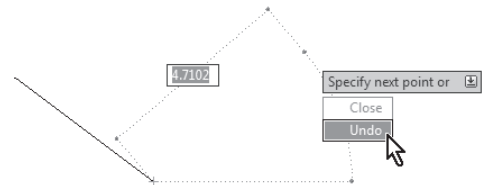
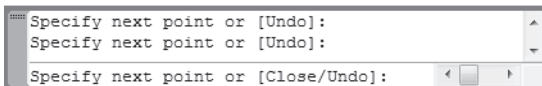
Commands can be entered in AutoCAD by several means, such as at the keyboard, through dynamic input, and by keyboard shortcuts. Let's look at each.

USING THE KEYBOARD

The original method for entering commands is through the keyboard. The earliest versions of AutoCAD did not rely on a mouse, which was expensive and rare at the time. (The technical editor notes that he used a joystick until it fell out of favor due to its association with games.) Every command had to be entered with the keyboard.

Over the years, menus, toolbars, ribbons, and right-click shortcut menus were slowly introduced to the user interface. Despite these changes to AutoCAD, you still need to deal with the keyboard, such as when entering coordinates or using commands that work only with the keyboard.

When typing commands at the keyboard, your input is displayed either in the command prompt area (illustrated below) or on the screen — depending on where the cursor is located.



Left: Entering commands in the command line.

Right: Entering commands on the screen (dynamic input).

After typing the command, you must press ENTER or the spacebar to activate the command. AutoCAD

recognizes these keystrokes in the command line:

Keystroke	Meaning
ESC	Cancels the current command.
⌫	Moves the cursor one character to the left.
⌵	Moves the cursor one character to the right.
HOME	Moves the cursor to the beginning of the command text.
END	Moves the cursor to the end of the line.
DEL	Deletes the character to the right of the cursor.
BACKSPACE	Deletes the character to the left of the cursor.
INS	Switches between insert and typeover modes.
⌶	Displays the previous line in the command history.
⌷	Displays the next line in the command history.
PGUP	Displays the previous screen of command text.
PGDN	Displays the next screen of command text.
CTRL+V	Pastes text from the Clipboard into the command line.
TAB	Cycles through command names.

Command Input Considerations

Before typing a command, the command line must be “clear.” That is, another command must not be in progress. If the ‘Command’ prompt is not clear, you can clear it by pressing ESC.

Specify next point or [Undo]: (Press ESC.)

Command:

Transparent commands are the exception. These commands are a selection of regular commands that, when prefixed with the ' (apostrophe) character, operate during other commands.

When you enter a command, AutoCAD doesn’t care if you use UPPERCASE or lowercase characters, or a combination of either. If you make a mistake typing on the command line, press the BACKSPACE key to correct, or edit the text on the command line using the keys listed in the table.

You can press ESC at any time. If you press ESC while an operation is in progress, it terminates the command.

Repeating Commands

To repeat the command you previously entered in AutoCAD, press ENTER (or the spacebar) at the ‘Command:’ prompt. As an alternative, right-click to display the cursor menu, and then select the **Repeat** option.

Some commands repeat automatically until you press ESC, but most commands do not. To force a command to repeat automatically, type **MULTIPLE** before entering the command name, as in:

Command: **multiple**

Enter command name to repeat: **circle**

This forces the CIRCLE command to repeat itself until you press ESC.

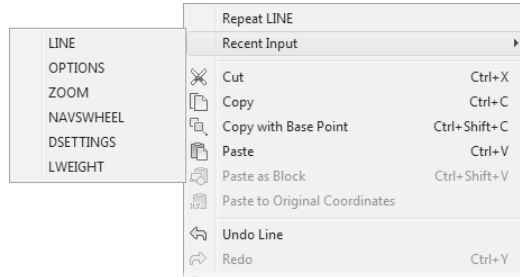
Recent Input

AutoCAD remembers the last twenty commands and options you enter at the ‘Command:’ prompt, or select from a menu or toolbar. Autodesk calls this “Recent Input.”

To access command history, press the ⌶ key at any time, except in a dialog box. AutoCAD shows the previous commands you entered. When you see the one you need, press the ENTER key, and the command is executed.

During commands, you can also press the $\mathbb{E}$ key. Instead of listing previous commands, AutoCAD shows you the coordinates you previously specified, whether entered at the keyboard or picked with the cursor.

If you prefer to see a list of the command history, right-click, and then select **Recent Input** from the shortcut menu. AutoCAD lists the previous commands and/or coordinate inputs in a submenu.



Dynamic Input

A second way to view commands typed at the keyboard is in the drawing area, as introduced in Chapter 1. Autodesk calls this “dynamic input,” because it displays tooltips, distances, angles, and tracking lines dynamically on the screen during drawing and editing commands. (*History:* The “dynamic” name goes back to Release 9, when Autodesk introduced the first dialog boxes to AutoCAD with the name of “dynamic dialogs.”)

There are three aspects to dynamic input, all of which you can turn on or off:

Dynamic Prompts — display commands and prompts in tooltips near the cursor.

Pointer Input — displays coordinates and angles in tooltips near the cursor. They can be edited.

Dimension Input — displays distances and angles in tooltips near the cursor. These appear only at the ‘Specify next point:’ prompt.

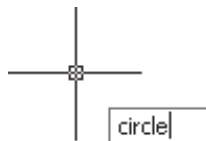
You turn on overall dynamic input like this:

1. On the status bar, turn on dynamic mode by clicking the  (or **DYN**) button.
2. Move the cursor into the drawing area, and then type commands, as described below.

To hide the command bar and force all command input to be dynamic, use the `COMMANDLINEHIDE` command. (Alternatively, press `CTRL+9` to toggle the display of the command prompt bar.)

Entering Commands

When entering commands during dynamic input, the command name appears in the drawing area near the cursor, such as “circle” in the figure below.

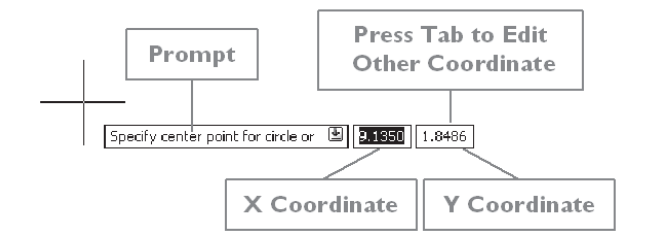


You can edit the command name by using the same keys as at the ‘Command:’ prompt. For example, you can press the $\mathbb{E}$ key to scroll through the names of previous commands, or use the `TAB` key to step through the names of commands that start with the same letter(s). Press `BACKSPACE` to erase the name, or `ESC` to exit the command.

If you enter an incorrect command name or invalid value, the input box is outlined in red.

Entering Coordinates

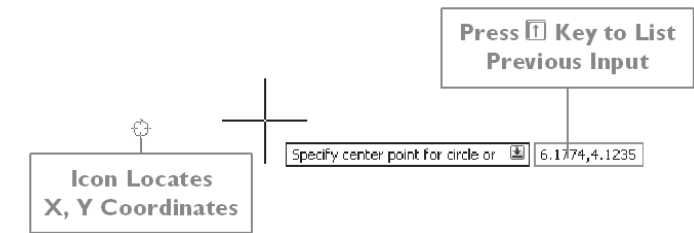
Upon entering the command name and pressing ENTER, the command’s prompts appear at the cursor. For the CIRCLE command, the first prompt is ‘Specify center point for circle:’. Pick a point in the drawing, or interact with the dynamic prompt, as described next.



A pair of numbers appears next to the prompt. In many cases, the number represents the x and y coordinates of the cursor’s current position. As the cursor moves, the values update.

The x coordinate of 9.1350 is highlighted, because it can be edited. To edit the y coordinate, press the TAB key; press the TAB key a second time to return to the x coordinate.

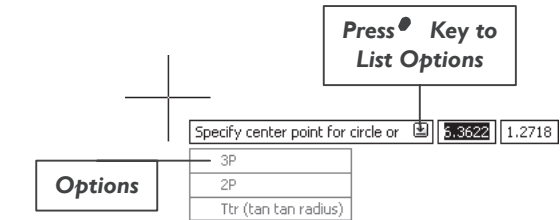
You can press the ↵ key repeatedly to cycle through earlier inputs. As you do, older x, y coordinates appear next to the prompt. In addition, an orange marker appears in the drawing showing the location of the coordinate.



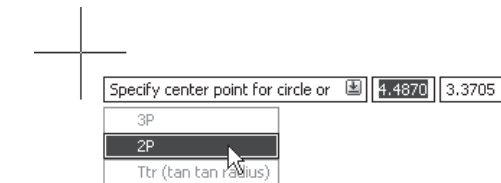
Press ENTER to fix the x, y-coordinate.

Selecting Options

Next to the prompt text is a small icon that looks like an arrow on a keycap.

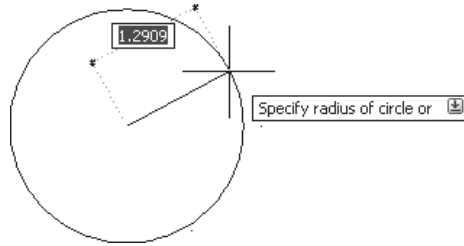


This is a reminder to press the ↵ key to see the available options. (When the arrow-key icon does not appear, the command has no options at this time.) To select an option, move the cursor to it, and then click, such as **2P** illustrated below. (The **2P** option draws two-point circles.)



Dynamic Dimensions

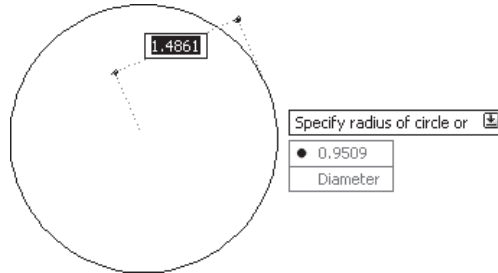
After placing the first point, AutoCAD displays the **CIRCLE** command's next prompt, 'Specify radius of circle:'. Also shown is the *dynamic dimension*, which displays the size of the circle's radius.



Technically, dynamic dimensions show the relative distance from the previous point to the current cursor location. The same distance is shown on the status bar, if relative coordinates are selected.

Selecting the Default Value

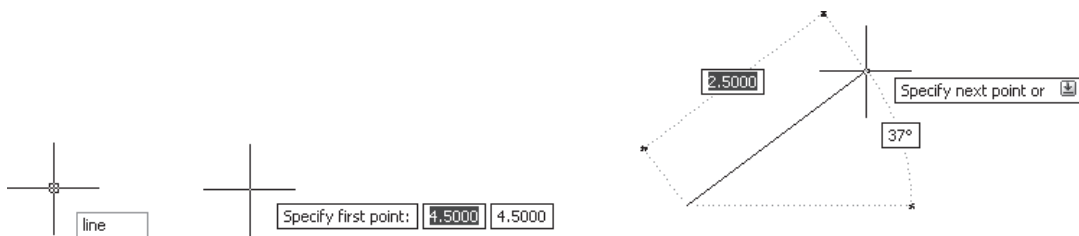
As before, you can press the **↵** key to select an option. This time, the previous radius (or diameter) is shown with a black dot next to it. The black dot, as in • 0.9509, indicates this is the default value. (This is the equivalent to seeing '<0.9509>:' in the command prompt area.)



To select the default, press **ENTER**, or pick the value with the cursor.

Dynamic Angles

Dynamic dimensions also display angles. (The angle was not shown for constructing the circle, because the angle doesn't matter for circles specified by a radius.) To see the angle, start the **LINE** command, and then pick two points. After picking the first point, notice that the dynamic angle appears:



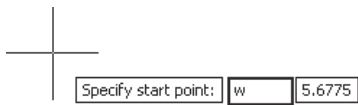
The angle of 37° is measured from the positive x axis.

Error Messages and Warnings

Dynamic input also displays error messages and warnings at the cursor.

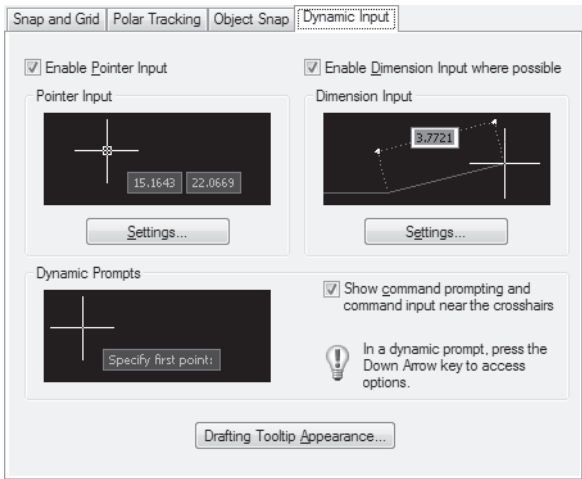


When you enter an incorrect value for coordinates, AutoCAD displays a red box around the problem, such as the “w” below:

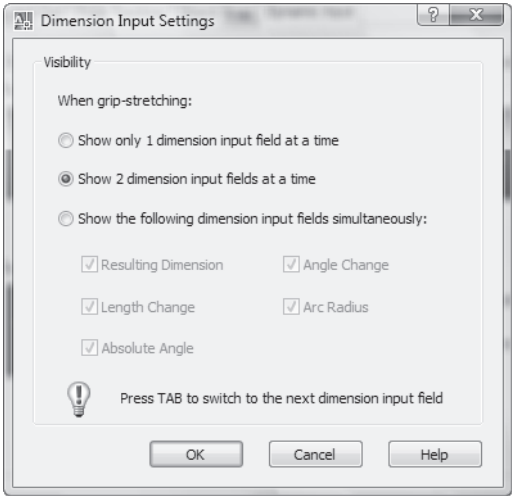
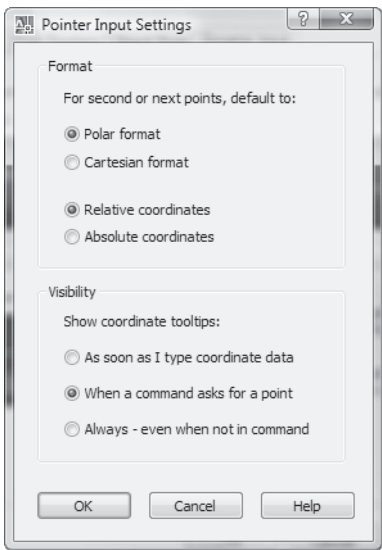


Modifying Dynamic Input

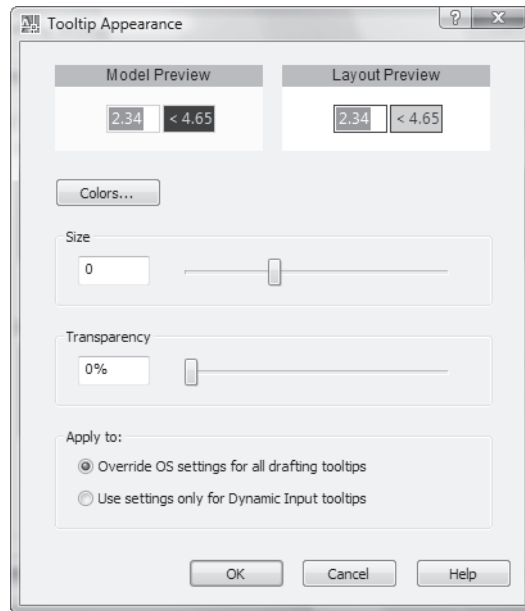
AutoCAD allows you to modify many aspects of dynamic input. To make changes, right-click the DYN button, and then select **Settings** from the shortcut menu.



Notice that you can toggle pointer input, dimension input, and command prompting. Click the **Settings** buttons to determine how and when dynamic input appears.



To change the color and size of the tooltips, click the **Drafting Tooltip Appearance** button.



Keyboard Shortcuts

In addition to accepting the full name of commands, AutoCAD allows you to type shortcuts. AutoCAD recognizes several types of shortcuts, including *aliases*, *function keys*, and *control keys*. The advantage to shortcuts is that you execute commands more quickly.

AutoComplete

AutoCAD has approximately a thousand commands and system variables, and it can be difficult to remember the correct spelling of each. For example, do you enter POLYLINE or PLINE? POLYGON or PGON? One way to deal with the confusion is to select command names from the menu; however, you have to know the menu in which the command resides. And the names and locations of commands in menus and ribbons change from release to release.

An alternative is to type the start of a command, such as “p,” and then press the TAB key. Each time you press TAB, AutoCAD displays the next command, system variable, or alias in alphabetical order. Autodesk calls this “AutoComplete.”

For example, enter **p**, and then press TAB:

Command: **p** (Press TAB.)

Command: **P** (Alias for PLOT command; press TAB.)

Command: **PA** (Alias for PASTESPEC command; press TAB.)

Command: **PAGESETUP** (Press ENTER to accept this command.)

Press **SHIFT+TAB** to go backwards through the list. For example, enter **p** and then press **SHIFT+TAB**. The next to appear is the **PURGE** command, and then the **PUCSBASE** system variable.

When you reach the command you need, press **ENTER** to execute it.

Function Keys

Function keys appear on the top row of your keyboard; they are all prefixed with the letter F (short for “function”). AutoCAD assigns meanings to many of the function keys:

Function Key	Meaning
F1	Calls up the help window.
F2	Toggles between the graphics and text windows.
F3	Toggles object snap on and off.
F4	Toggles tablet mode on and off; you must first calibrate the tablet before toggling tablet mode.
F5	Switches to the next isometric plane when in iso mode; the planes are displayed in order of left, top, right, and then repeated.
F6	Toggles dynamic UCS on and off.
F7	Toggles grid display on and off.
F8	Toggles ortho mode on and off.
F9	Toggles snap mode on and off.
F10	Toggles polar tracking on and off.
F11	Toggles object snap tracking on and off.
F12	Toggles dynamic input mode.
CTRL+F4	Closes the current drawing.
ALT+F4	Exits AutoCAD.
ALT+F8	Displays the Macros dialog box.
ALT+F11	Starts the Visual Basic for Applications editor.
ALT+TAB	Switches to the next application.
CTRL+TAB	Switches to the next drawing (window).

Aliases

Aliases are abbreviations of full command names. For example, the alias of the LINE command is **L**. To execute the LINE command more quickly, press **L**, followed by ENTER. AutoCAD executes the LINE command just as if you had entered the full command name. Examples of common aliases include:

Alias	Command
a	ARC draws arcs.
aa	AREA finds the area.
adc	ADCENTER opens the Design Center.
b	BLOCK creates blocks (symbols).
c	CIRCLE draws circles
co	COPY copies objects.
h	HATCH opens the Hatch dialog box.
i	INSERT inserts blocks.
l	LINE draws lines.
la	LAYER opens the Layers palette.
le	LEADER draws leaders.
m	MOVE moves objects.
t	TEXT places text in drawings.
z	ZOOM magnifies the drawing.

There are many other command aliases, and you can create your own. The full list provided with AutoCAD is described in Appendix B.

Other Keys

AutoCAD uses the **SHIFT** key to override drafting settings temporarily. For some, there are two shortcuts, one for left-hand and another for right-handed mouse users.

You use control-key shortcuts by holding down the **CTRL** key, and then pressing another key. You may already be familiar with some of them, such as **CTRL+C** to copy to Clipboard, and **CTRL+Z** for undo. AutoCAD displays its control-key shortcuts in the drop-down menus, next to the related command. These keystrokes can also be customized (changed) with the CUI command.

See Appendix B for the complete list of keyboard shortcuts.

USING MOUSE BUTTONS

AutoCAD recognizes as many as sixteen buttons on mice and digitizer pucks. Most mouse devices, of course, are limited to two, three, or four buttons, but digitizer pucks commonly have four, twelve, or sixteen buttons.



By default, AutoCAD defines the first ten buttons found on input devices, as follows:

Button #	Mouse Button	Meaning
1	Left	Selects commands or objects.
2	Right	Displays shortcut menus.
3	Center	Pans the drawing (when MBUTTONPAN = 1).
SHIFT+1	SHIFT+Left	Toggles cycle mode.
SHIFT+2	SHIFT+Left	Displays object snap shortcut menu.
CTRL+2	CTRL+Right	Displays object snap shortcut menu.
...	Wheel	Zooms or pans.
Button #	Meaning	
4	Cancels command.	
5	Toggles snap mode.	
6	Toggles orthographic mode.	
7	Toggles grid display.	
8	Toggles coordinate display.	
9	Switches to next isometric plane.	
10	Toggles tablet mode.	

Buttons 11 through 16 are not predefined by AutoCAD. It is possible to change the meaning of all the buttons except the first (left) button, which is always the Select button. (The Windows mouse driver can reverse the meaning of the left and right buttons for left-handed users.)

Left Mouse Button

Earlier, I noted that you can pick points in the drawing with the cursor. By moving the mouse, you control the movement of the cursor. To pick a point, press the mouse's left button.

Right Mouse Button

In many cases, pressing the right mouse button (when inside the AutoCAD window) displays a *context-sensitive shortcut menu*. "Context-sensitive" means that the content of the menu changes, depending on *where* you right-click and *which* command is in effect.

The shortcut menu (cursor menu) appears at the cursor when you press the right-mouse button.

Roller Wheel

The wheel on a mouse can zoom and pan the drawing without invoking the ZOOM and PAN commands.

To zoom in or out, roll the wheel forward (zoom in) and backward (zoom out). To zoom the drawing to its extents, double-click the wheel button.

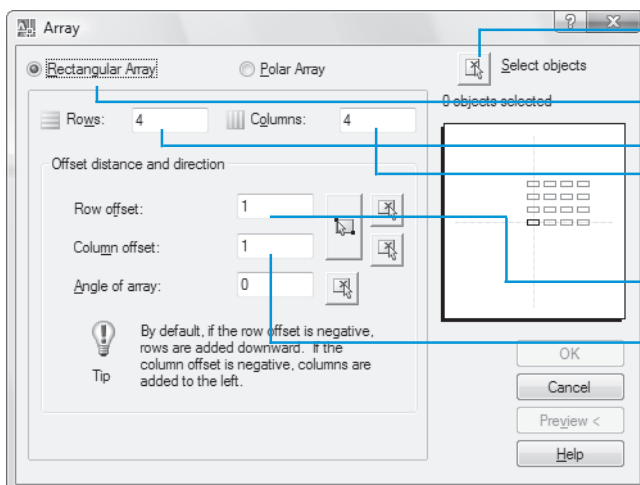
To pan about the drawing, hold down the wheel and drag the mouse.

In some cases, you might prefer that the wheel act as a button. You can do this by setting the value of the MBUTTONPAN system variable to 0. Now when you click the wheel, it acts like the middle button of a three-button mouse. Some mouse brands come with more than three buttons. This allows you to access additional AutoCAD functions.

USING DIALOG BOXES

Many commands require you to specify options in dialog boxes. The benefit of dialog boxes is that they display all your options at the same time, unlike the command-line, which typically displays one option at a time. The dialog box and the command-line for creating arrays are compared below:

Command: **array**



Command: **-array**

Select objects: (Select one or more objects.)

Select objects: (Press ENTER.)

Enter the type of array [Rectangular/Polar] <R>: (Press ENTER.)

Enter the number of rows (---) <1>: 3

Enter the number of columns (|||) <1>: 4

Enter the distance between rows or specify unit cell (---): 1

Specify the distance between columns (|||): 1

The drawback to dialog boxes is that they obscure drawings. On the other hand, dialog boxes present options grouped logically; command-line prompts guide you through the steps needed to complete the action, but lack the previews and hints found in dialog boxes. The command-line steps you through options serially; dialog boxes allow you to jump to options directly

A number of commands can display either a dialog box or prompts on the command line. There are several reasons for this. One is historical: until Release 9, AutoCAD didn't use dialog boxes at all,

so all commands had to be entered at the command prompt. Also, many commands don't need a dialog box, such as drawing and editing commands like **LINE** and **COPY**. A third reason is that you may prefer using the command line, because it can be faster than using dialog boxes. For example, I find it faster to type the **-VPOR**TS command and its options than to use the Viewports dialog box. A final reason is technical: script files and AutoLISP routines cannot control dialog boxes, and so must use the command-line version. There are two ways to suppress dialog boxes.

Hyphen Prefix

Prefixing some commands with a hyphen (-) suppresses the related dialog box, displaying prompts at the command line instead. For example, the **ARRAY** command normally displays the Array dialog box:

Command: **array**
(AutoCAD displays dialog box, shown above.)

Prefixing the command with a hyphen displays the command line options:

Command: **-array**
Select objects: *(Select one or more objects.)*
Select objects: *(Press ENTER.)*
(Command continues with the prompts listed above.)

Not all commands have the hyphen option; the following table lists some of those that do:

Commands	Hyphenated
ARCHIVE	-ARCHIVE
ARRAY	-ARRAY
ATTDEF	-ATTDEF
ATTEDIT	-ATTEDIT
ATTEXT	-ATTEXT
BEDIT	-BEDIT
BLOCK	-BLOCK
BOUNDARY	-BOUNDARY
COLOR	-COLOR
DIMSTYLE	-DIMSTYLE
HATCH	-HATCH
HATCHEDIT	-HATCHEDIT

There are some curious exceptions: **-PARTIALOPEN** is the true name, while **PARTIALOPEN** is its alias. Similarly, **-SHADEMODE** is the true name, while **SHADEMODE** is an alias for the **VSCURRENT** command.

System Variables

Whether or not certain commands display dialog boxes is controlled by system variables. *System variables* store the current state of AutoCAD. Here are a few common examples.

FileDia

When the **FILEDIA** system variable is set to 0, AutoCAD suppresses the display of file-related dialog boxes, such as those associated with the **NEW**, **OPEN**, **SAVEAS**, and **VSLIDE** commands. Prompts are displayed at the command line:

Command: **filedia**
Enter new value for FILEDIA <1>: **0**
Command: **open**
Enter name of drawing to open: *(Enter path and name of .dwg drawing file.)*

To force the display of the dialog box, type the tilde (~) character when AutoCAD prompts for the

file name:

Command: **open**

Enter name of drawing to open: ~

(AutoCAD displays the Select File dialog box.)

Expert

When the **EXPERT** system variable is set to a value other than zero, it suppresses warning dialog boxes displayed by AutoCAD. Examples include “About to regen, proceed?” and “Really want to turn the current layer off?”

AttDia

When the **ATTDia** system variable is set to 0, the **INSERT** command displays prompts for attribute data at the command line; when set to 1, a dialog box is displayed instead. (Curiously, though, in this age of dialog boxes and palettes, 0 continues to be the default.)

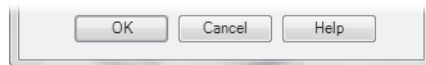
Alternate Commands

In a very few cases, AutoCAD uses one command name for the command line, and a different one for dialog boxes. Examples include:

Command Line	Dialog Box
FILEOPEN	OPEN
CHANGE	PROPERTIES
UCS	UCSMAN

Controlling Dialog Boxes

When a dialog box opens, the arrow cursor replaces the crosshair cursor. Use the cursor to choose options in the dialog box. Then choose the **OK** button to accept the changes and dismiss the dialog box.



To dismiss the dialog box without affecting the drawing, choose **Cancel**.

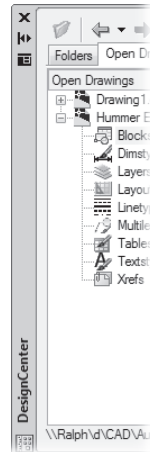
Almost all dialog boxes contain a **Help** button. Selecting the button displays a help box that explains the purpose and use of the dialog box. Many also have a **?** button in the upper-right corner: click it, and then click an item in the dialog box for a context-sensitive explanation.

Some dialog boxes have an **Apply** button to apply changes you made, yet keep the dialog box open. Others have a **Preview** button that temporarily dismisses the dialog box, so that you can see the changes.

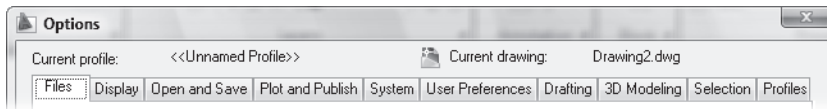
You can move dialog boxes around the screen by dragging them by their *title bar*. This means that you position the cursor over the title bar, and holding down the left mouse button, move the dialog box. Moving the dialog box is useful if you need to see objects under the box.

Most dialog boxes are *modal*, which means you must dismiss them before you can continue working in AutoCAD. A few are *non-modal*, which means they hover on your Windows desktop as you continue working in AutoCAD; these are called “palettes.”

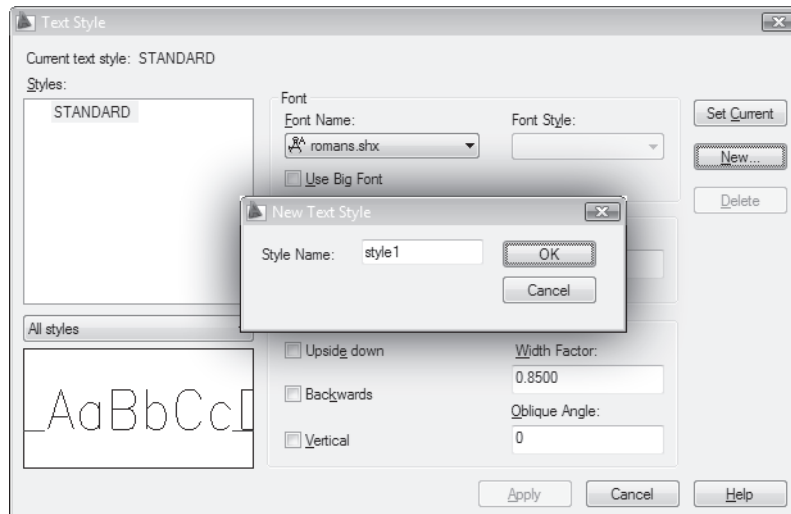
An example is the DesignCenter palette. To dismiss a non-modal dialog box, click the small **x** in the upper corner.



Other dialog boxes contain *tabs*, which allow a single dialog box to display a lot of information, separated into similar-looking boxes. For example, the **OPTIONS** command displays a dialog box with nine tabs!



Some dialog boxes contain additional or sub-dialog boxes. If a sub-dialog box is displayed, you must choose **OK** or **Cancel** to return to the original dialog box in order to continue. In the figure below, New Text Style is the sub-dialog box displayed by selecting the New button.



Buttons in Dialog Boxes

Buttons are used to select items. Select buttons with the cursor or by pressing keys on the keyboard. To move from button to button, press the **TAB** key.

- The default button is the one with the heavy border. Pressing **ENTER** is like choosing the button with the cursor. In the figure above, the **OK** button has the heavy border.
- Three periods (...) after a button's name opens a sub-dialog box. An example is the **New...** button in the dialog box illustrated above.
- AutoCAD uses several different styles of button to expand or temporarily dismiss the dialog box. For example, the **Preview <** button lets you access the drawing (illustrated below, left).



Buttons with a round arrow button expand the dialog box, as shown above (center). Alternatively, the dialog box may have a **Show Details** button (above, right)

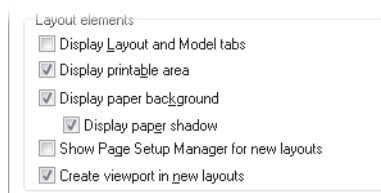


- If the text on the button is **gray**, the button's function is not available at this time, such as the **Preview** and **OK** buttons above.

The following is a summary of the buttons used in dialog boxes.

Check Boxes

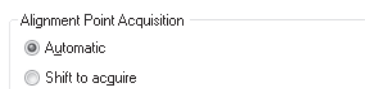
Check boxes *toggle* options, which means the option is either on or off.



- ☒ A check mark indicates the option is selected (or turned on).
- ☐ No check mark means the option is turned off.

Radio Buttons

Round radio buttons select one option out of several. Only one option is active at a time.



- Ⓐ The black dot in the button indicates it is selected or turned on.
- No dot indicates it is unselected or turned off.

Select Objects Button

Some dialog boxes allow you to pick objects or points in the drawing. When you click the  **Select Objects** button, AutoCAD temporarily dismisses the dialog box, and prompts you to pick the points, such as:

Pick a point: *(Pick a point.)*

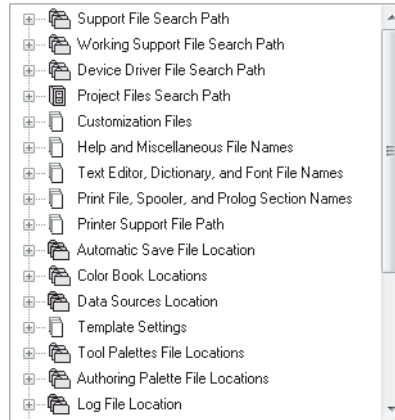
Pick a point (or press ENTER to accept): *(Press ENTER to return to the dialog box.)*

Once done, the dialog box returns.

List Boxes

List boxes contain a list from which you choose one or more items. When the list is too long for the box, a scroll bar appears automatically, allowing you to scroll to choices not visible. To choose more than one item, hold down the **CTRL** key as you select items; not all list boxes allow this.

Some list boxes can be dragged larger with the cursor.

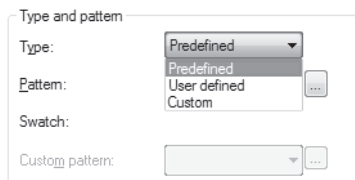


Scroll Bars

Some list boxes contain more entries than can be displayed at a time. Scroll bars exist on some list boxes to bring additional items or options into view by scrolling up or down. To display another entry below the current field of view, click on the down arrow of the scroll bar.

Droplists

Droplists normally display a single item. They have an arrow button; when you click the button, a list “drops down” to show the entire list. An example is shown below from the Hatch dialog box.



You can also move through the scroll list by clicking on the bar, and while continuing to hold the mouse button down, sliding the bar up or down with the cursor. When you click again, the entries are redisplayed at the new location. Note that the position of the scroll bar is relative to the position of the displayed items. Thus, if there are many items in the list, a relatively small movement of the scroll bar moves by several items.

Text Edit Boxes

Text edit boxes contain a single line of text (or, in some cases, a paragraph of text) to be edited. To edit the text, click in the box. A cursor bar appears at the text. You move the cursor bar with the arrow keys on the keyboard.



THE DRAFT-EDIT-PLOT CYCLE

Once you set up a drawing, working with AutoCAD involves a cycle, consisting of drafting, editing, and plotting. You draft drawings, edit, and then plot them. This book is arranged in a similar manner:

DRAFTING

Drafting consists of placing and constructing objects in the drawing. AutoCAD comes with a set of objects, from which everything else in the drawing must be made:

Objects	Drawn with Command(s)
Points	POINT
Lines	LINE, TRACE, PLINE, <i>and</i> 3DPLINE
Parallel lines	MLINE
Construction lines	RAY <i>and</i> XLINE
Circles	CIRCLE <i>and</i> DONUT
Ellipses	ELLIPSE
Arcs	ARC <i>and</i> PLINE
Elliptical arcs	ELLIPSE
Regular Polygons	POLYGON
Rectangles	RECTANG
Splines	SPLINE <i>and</i> HELIX
Irregular boundaries	BOUNDARY
Revision clouds	REVLOUD
3D polymeshes	AI_BOX, AI_DOME, AI_DISH, AI_PYRAMID, AI_CONE, AI_CYLINDER, AI_SPHERE, AI_TORUS, AI_WEDGE, AI_MESH, <i>and</i> 3DFACE
3D meshes	MESH, EDGESURF, SURFTAB, RULETAB, TABSURF
3D surfaces	PLANESURF, LOFT, SWEEP, THICKEN
3D solids	BOX, CONE, CYLINDER, SPHERE, TORUS, PYRAMID, POLYSOLID, <i>and</i> WEDGE
Text	TEXT, MTEXT, <i>and</i> FIELD
Tables	TABLE
Dimensions	<i>All commands starting with</i> DIM, <i>plus</i> LEADER, QDIM, QLEADER, TOLERANCE, <i>and</i> MLEADER
Parametrics	PARAMETERS, DIMCONSTRAINT, <i>and</i> AUTOCONSTRAIN
Constraints	GEOMCONSTRAINT

Constructing Objects

When you cannot draw an object with one of AutoCAD's commands, you must *construct* it. For example, AutoCAD has no commands for drawing right-angle triangles and irregular polygons, so you construct them with the LINE or PLINE commands. In addition, AutoCAD has a collection of commands to construct copies of objects:

Copy Method	Copied with Command(s)
Copy objects	COPY
Copy blocks	INSERT, XBIND <i>and</i> ADCENTER
Offset copies	OFFSET
Mirror copies	MIRROR <i>and</i> MIRROR3D
Array copies	ARRAY <i>and</i> 3DARRAY
Array as block	MINsert
Copy faces, edges	SOLIDEDIT, FLATSHOT <i>and</i> IMPRINT
Copy along paths	DIVIDE <i>and</i> MEASURE

BLOCKS AND TEMPLATES

One of Autodesk’s cofounders, John Walker, said that you should never have to draw the same object twice in CAD. The best way to reuse drawing details is to turn them into *blocks*. Blocks can be used repeatedly in drawings, shared between drawings and offices, and can also hold database information, called “attributes.”

Sometimes you start drawings with *templates*, other drawing files that already contain previously-created settings and objects. The most common templates consist of a drawing border, title block, and perhaps layers. Many firms also use templates for standard drawings of details.

EDITING

Editing consists of modifying objects in the drawing, correcting mistakes, moving objects, changing your mind, your boss changing your mind, and so on.

Editing Method	Edited with Command(s)
Move objects	MOVE, STRETCH, ALIGN, and 3DMOVE
Rotate objects	ROTATE, ROTATE3D, and 3DROTATE
Resize objects	SCALE
Stretch objects	STRETCH, LENGTHEN, and EXTEND
Mirror objects	MIRROR
Erase objects	ERASE
Trim objects	TRIM, BREAK
Fillet objects	FILLET
Chamfer objects	CHAMFER
Undo editing	U, UNDO, REDO, and MREDO
Edit splines	SPLINEDIT
Edit multilines	MLEDIT
Edit text	EDITTEXT
Change text size	SCALETEXT and STYLE
Align text	JUSTIFYTEXT and STYLE
Edit attributes	EATTEDIT and ATTIPEDIT
Edit polylines	PEDIT
Edit 3D solids	UNION, INTERSECT, SUBTRACT, and SOLIDEDIT
Reduce objects	EXPLODE and XPLODE
Change properties	CHANGE and PROPERTIES
Reverse directions	REVERSE

In addition to using commands to edit, you can edit objects directly. For example, double-clicking most objects brings up the Properties palette, which lets you change their properties; double-clicking text brings up the appropriate text editor.

Single-clicking objects produces *grips* (handles), allowing you to stretch, move, copy, rotate, mirror, and resize them directly with the cursor.

In addition to editing objects, AutoCAD has a collection of commands that *construct* objects:

Construction Type	Constructed with Command(s)
Hatch patterns	HATCH, GRADIENT, and HATCHEDIT
Filled areas	SOLID, and HATCH
Blank areas	WIPEOUT
3D revolutions	REVOLVE
3D extrusions	EXTRUDE
3D solids	INTERFERE, SLICE, SECTION, IMPRINT, PRESSPULL, THICKEN, SWEEP, and LOFT
3D meshes	MESHSMOOTH

PLOTTING

When your drawings are complete, you print them on paper (a process called “plotting”), or send them through the Internet, called an “e-plot,” short for “electronic plot.” When plots are created before the drawing is complete, they are called “check plots.”

AutoCAD’s **PREVIEW** command displays plots before they are committed to paper, while the **PLOTSTAMP** command can be used to label plots with the drawing file name, date and time of plot, and other information.

AutoCAD supports two kinds of plots: *color based* and *style based*.

- **Color-based** plotting is the older method, whereby the colors of objects determine the pens (or inkjets) with which they are plotted.
- **Style-based** plotting applies *styles* to layers and individual objects; these styles affect every aspect of how objects can be plotted: color of lines, width of lines, screen percentage, style of end caps, and even whether the objects are plotted at all.

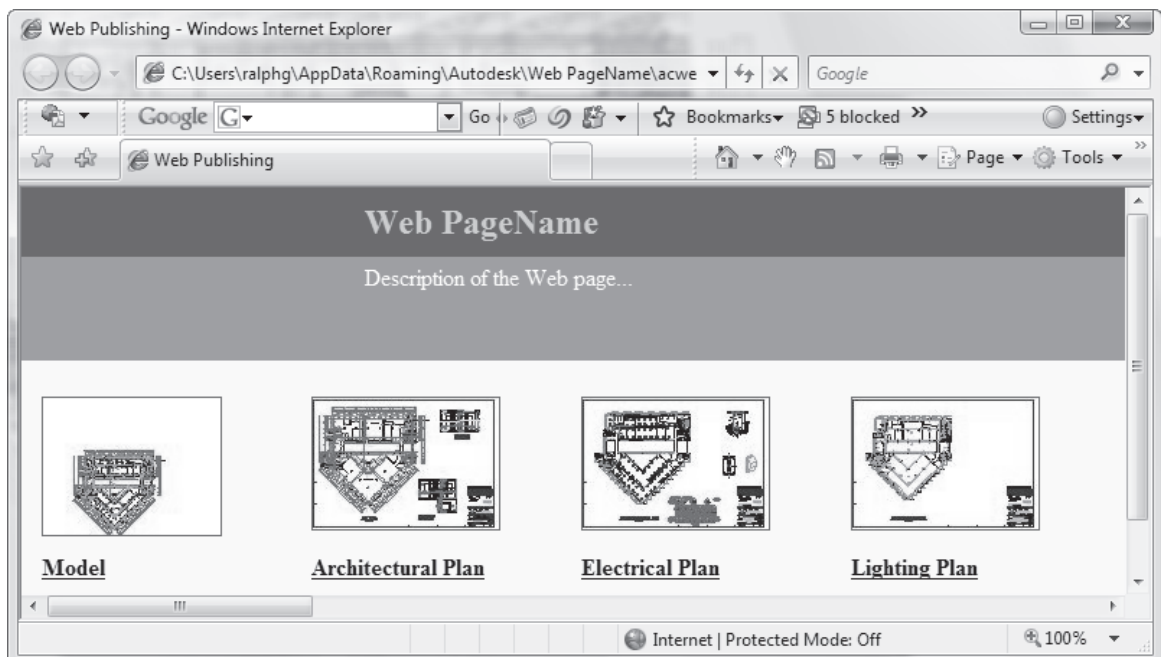
ePlots

In addition to plotting drawings on paper, you can also plot drawings to file. This lets you send copies of drawings without sending the original *.dwg* files. These are sometimes called “electronic plots.” The *.dxf* and *.pdf* files look just like the originals, but cannot be edited. Save drawings in DWF format (short for “design web format”) with the **EXPORTDWF** command or in PDF format (short for “portable document format”) with the **EXPORTPDF** command.

To allow you to view *.dxf* files on your computer and over the Internet, Autodesk provides the Design Review software with AutoCAD. Non-AutoCAD users can download the software free from www.autodesk.com/designreview.

To view *.pdf* files, most computers include the Acrobat Reader from Adobe; you can download it free from www.adobe.com/acrobat. Viewing software allows you to zoom in and out, select predefined views, and print drawings.

The **PUBLISHTOWEB** command generates Web pages and images of the drawings, while the **3DDWF** command creates 3D versions of DWF format files.



PROTECTING DRAWINGS

To protect drawings, AutoCAD includes *password* protection and digital signatures. When you save drawings with passwords (through the SECURITYOPTIONS command), they cannot be opened unless the correct passwords are entered.

Be careful: If you forget the password, the drawing is rendered useless. In this regard, it's best not to use passwords at all.

A different level of security is the *digital signature*. When drawings are “signed” digitally, a warning appears after they have been edited or altered. This alerts you that the drawings may not be originals; the system does not, however, tell you *what* has been changed. You must purchase a digital ID from a certificate authority in order to apply digital certificates.

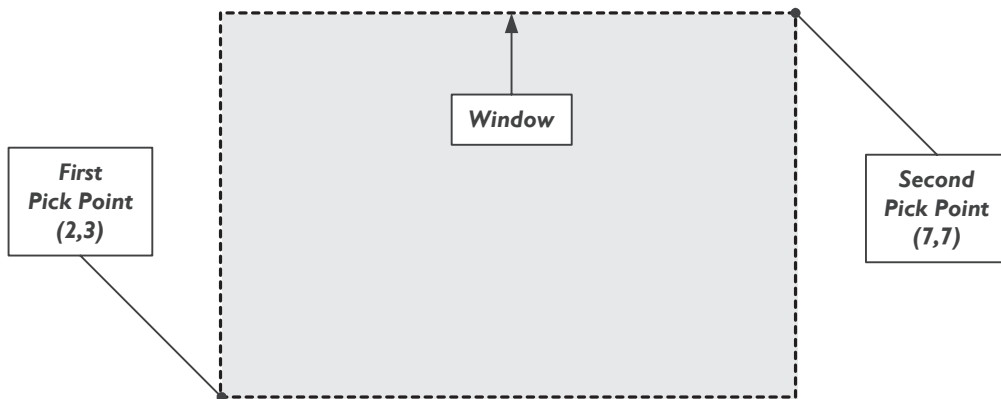
SCREEN POINTING

You enter points, distances, and angles in two ways: by “showing” AutoCAD the information on the screen, or by entering coordinates and angles at the keyboard.

Picking two points in the drawing could indicate, for example, a distance, an angle, or both — depending on the command and what it expects as input. With AutoCAD, you press the left mouse button to pick points in the drawing.

Showing Points by Window Corners

Some commands require input of both a horizontal and a vertical displacement. This is shown by a rectangle on the screen, called the “window.” You pick a point in the drawing, and then move the cursor to form the rectangle. AutoCAD determines the x and y distances by measuring from the lower left corner to the upper right corner.



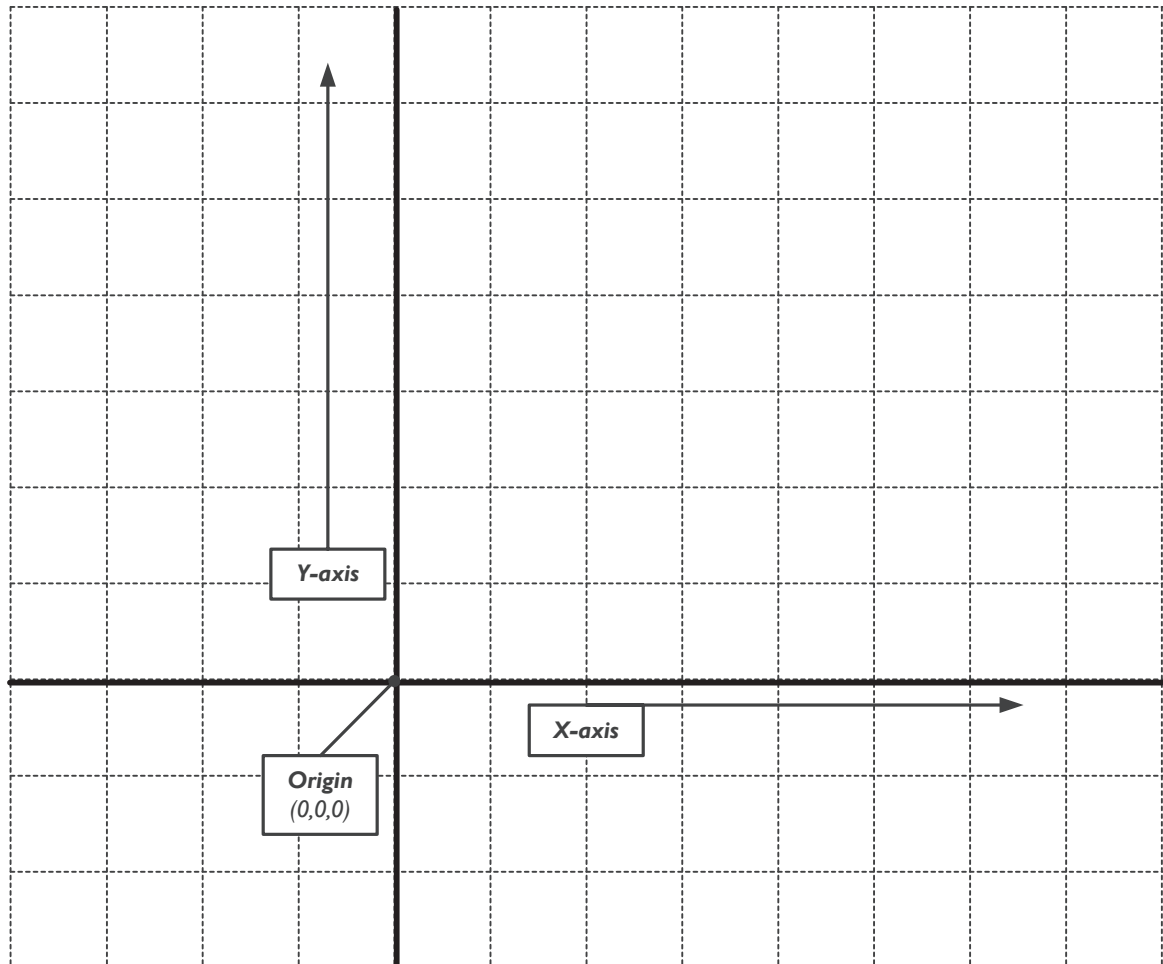
For example, the window in the figure above starts at (2,3) and rises to (7,7). To AutoCAD, this indicates a displacement of (5,4) by using subtraction:

$$\text{Horizontal displacement (x)} = 7 - 2 = 5$$

$$\text{Vertical displacement (y)} = 7 - 3 = 4$$

COORDINATE SYSTEMS

AutoCAD works with *coordinate* systems. Coordinates locate objects in the drawing. (Technically, AutoCAD does not record lines and other objects, but rather their points and properties.) All 2D drafting is performed on the x,y-plane; for 3D drafting, you can redefine this plane to any orientation in space through UCSs.



AUTOCAD COORDINATES

AutoCAD works with several coordinate systems, most of which measure distances relative to the *origin*, which is located at 0,0,0:

- **Cartesian** define 2D and 3D points by two or three distances along the x, y, and (sometimes) z axes.
- **Polar** define 2D points by a distance and an angle.
- **Cylindrical** define 3D points by two distances and an angle.
- **Spherical** define 3D points by a distance and two angles.
- **Relative** define 2D and 3D distances (and optionally angles) relative to the last point.
- **User-defined coordinate systems (UCS)** define an x,y-plane anywhere in 3D space.

You enter coordinates in the format of your choice: decimal (metric), engineering (decimal inches), architectural (Imperial units), fractional (inches only), and scientific notation (exponents). Accuracy ranges from zero to eight decimal places (whole inches to 1/256th of an inch).

Angles can be entered in degrees-minutes-seconds (360 degrees in a circle), radians (2π), grads (400), and in surveyor's units (NdE).

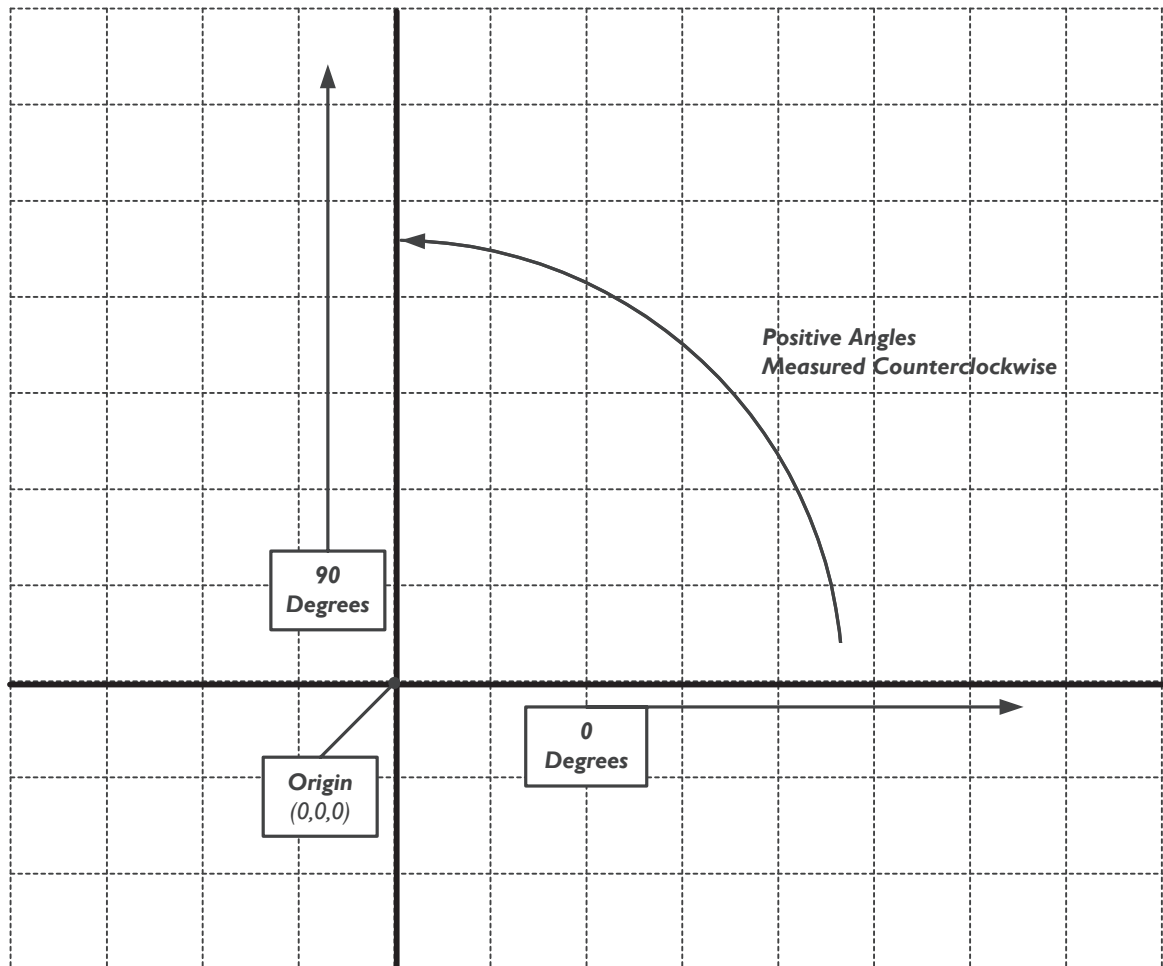
In new drawings, angles are measured counterclockwise, with 0 degrees pointing from the origin to the East, or 3 o'clock. Using this convention, 90 degrees points North (12 o'clock), 180 degrees points West (9 o'clock), and 270 degrees points South (6 o'clock).

To measure an angle clockwise, use negative angles. Prefix the angle with a dash, as in -45 degrees. Because there are 360 degrees in a circle, -45 degrees is the same as 315 degrees.

In the same way, you specify negative distances by prefixing numbers with the dash. For example, -2,-3 means two units in the negative x-direction, and three units in the negative y-direction.

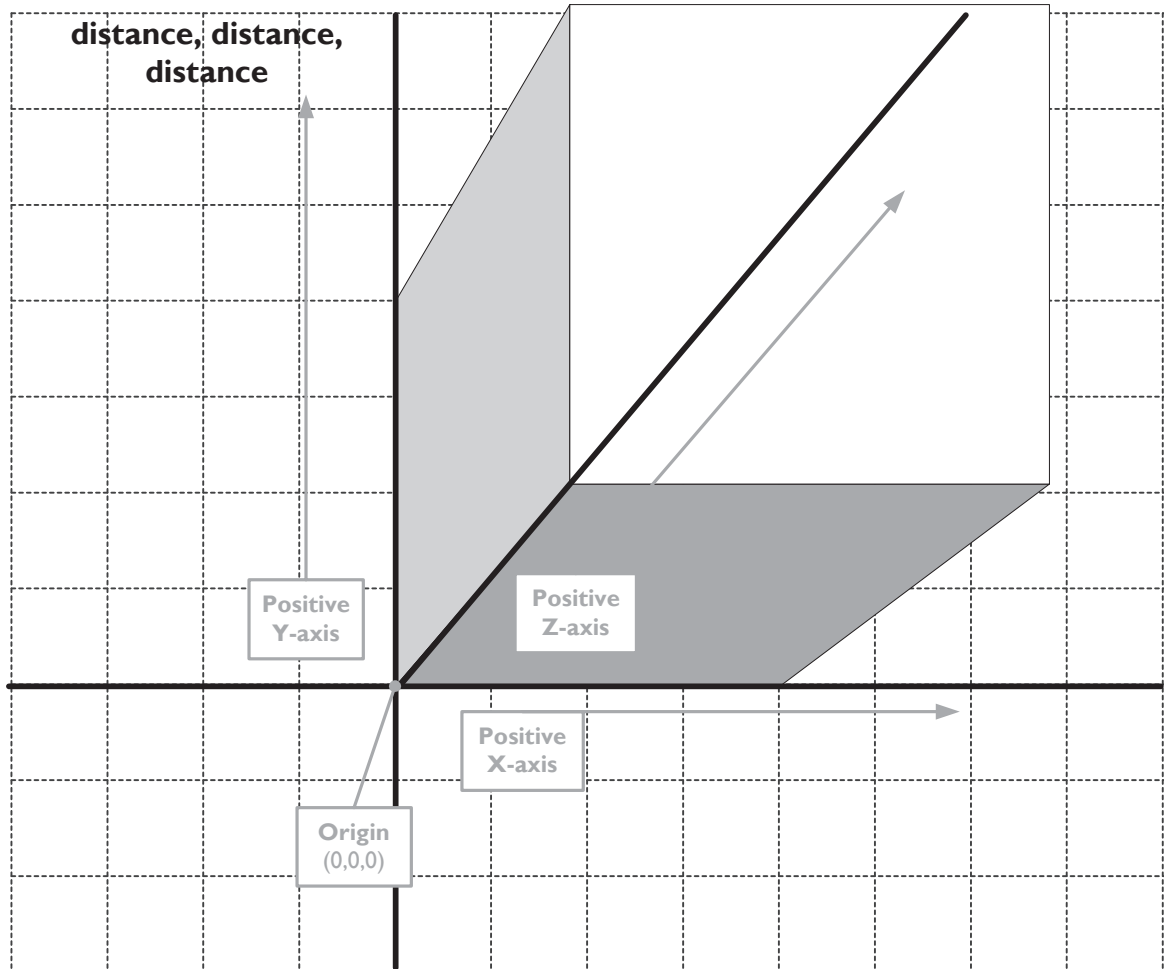
Recall that *absolute* coordinates are measured relative to the origin (0,0,0), while *relative* coordinates are measured relative to the last point. Similarly, absolute angles are measured from the x-axis counterclockwise, while relative angles are measured from the value stored in the LASTANGLE system variable.

If nothing else, AutoCAD is flexible in its measurement systems. For example, surveyors can change the UNITS command so that angles are measured clockwise and 0 degrees points in any direction.



Cartesian Coordinates

Cartesian coordinates define 2D and 3D points by measuring two or three distances: each distance is along one of the three axes: x, y, and z, either positive or negative. In AutoCAD, the x-direction is horizontal, at 0 degrees; positive x is measured to the right. The y-direction is vertical along 90 degrees; positive y is measured upward. The z-direction comes out of the screen; positive z points at your face.



To draw a line using Cartesian coordinates with x,y (2D) and x,y,z (3D) coordinates:

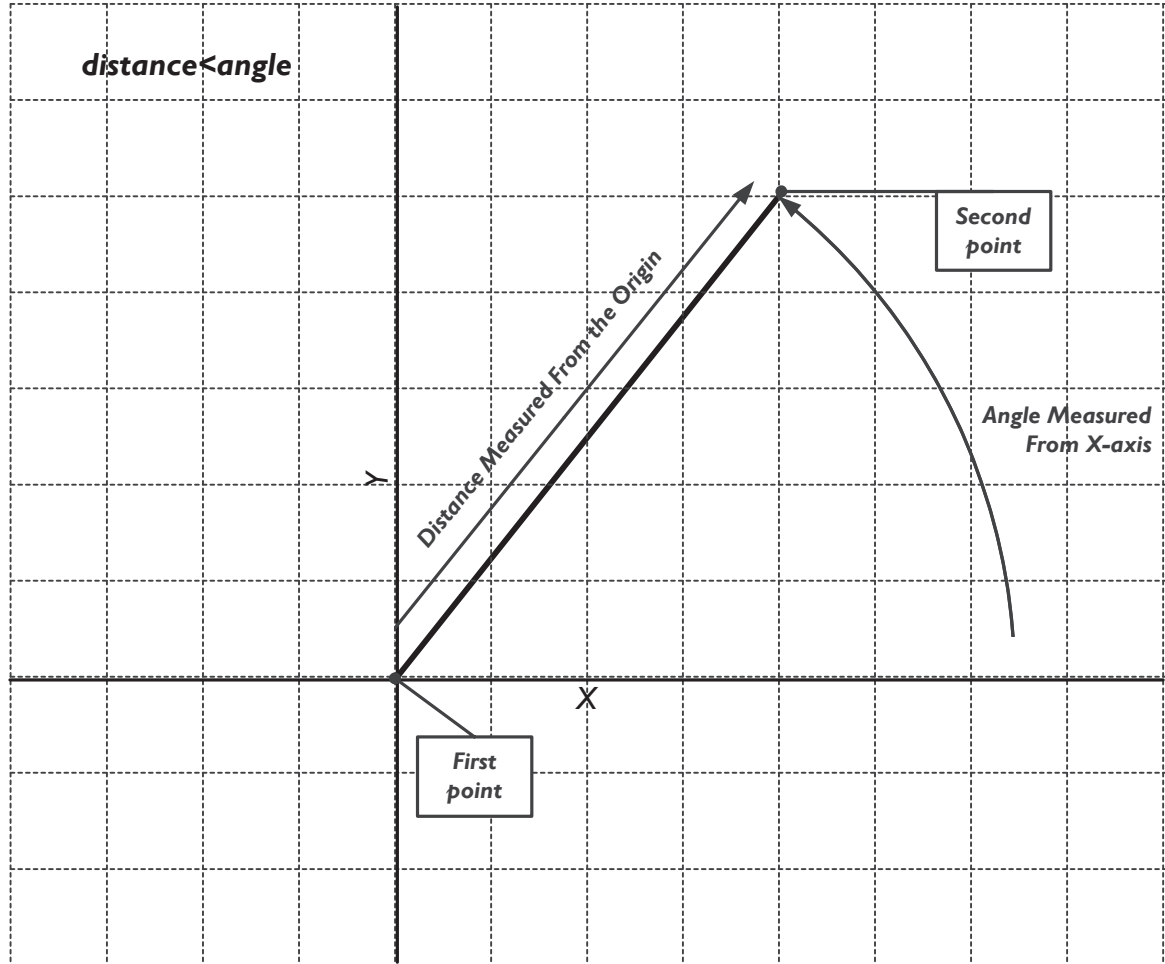
Command: **line**

Specify first point: **1,2**

Specify next point or [Undo]: **3,4,5**

Polar Coordinates

Polar coordinates define 2D points by measuring one distance and one angle in the x,y-plane. The distance is the straight-line distance measured from the origin to the point, while the angle is measured from 0 degrees. To indicate the angle to AutoCAD, you prefix the angle with an angle bracket (<), such as <45 for 45 degrees.



To draw a line using polar coordinates, all point are relative to the origin (0,0):

Command: **line**

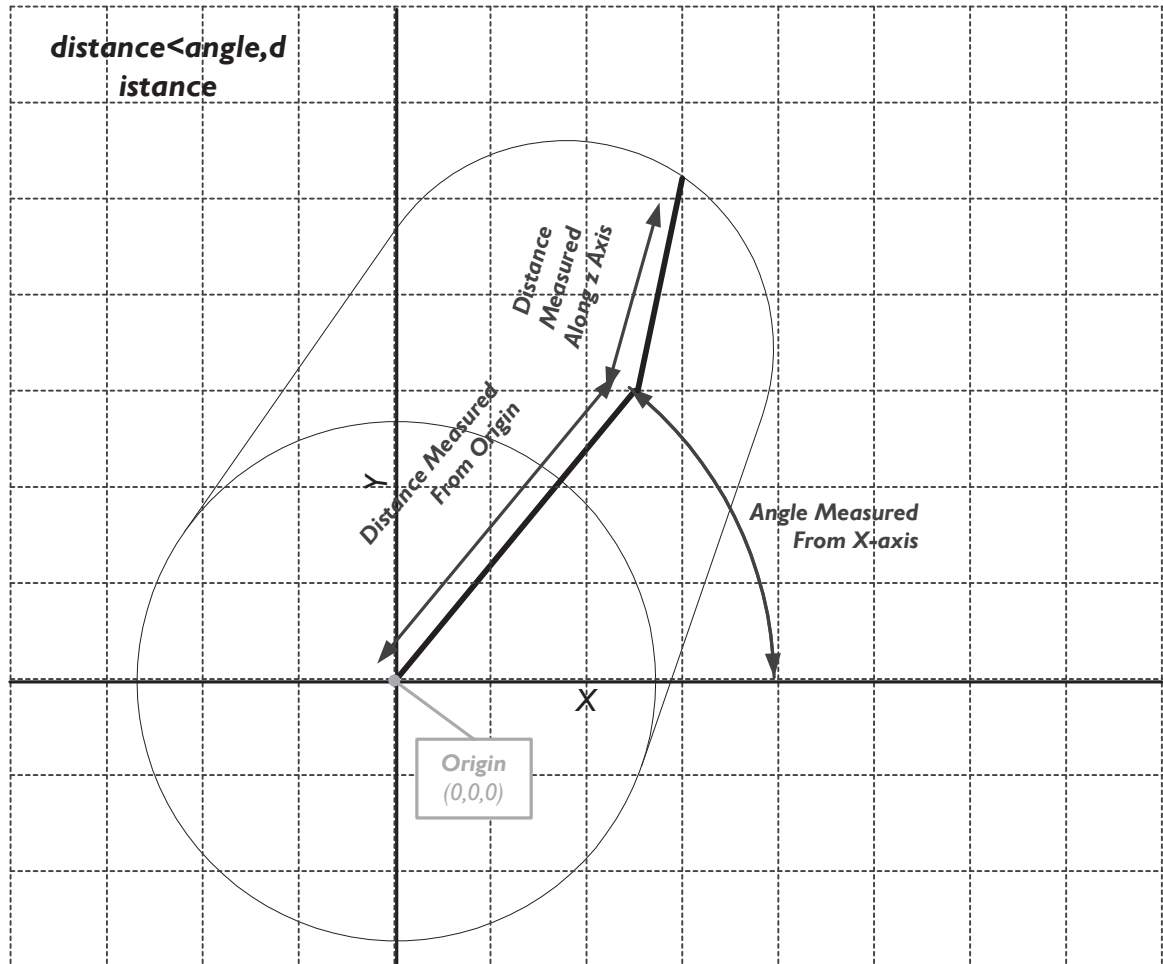
Specify first point: **1<23**

Specify next point or [Undo]: **4<55**

Cylindrical Coordinates

Cylindrical coordinates define 3D points by measuring two distances and one angle in 3D space. They are like polar coordinates, but with the z-distance added. The distances are measured from the origin to the point, while the angle is measured from 0 degrees.

The first distance lies in the x,y-plane; the second distance measures the distance along the z-axis. To indicate the measurement to AutoCAD, you use this notation: 1<23,4 — where 1 is the polar distance, <23 is angle from 0 degrees, and 4 is the height in the z direction.



To draw a line using cylindrical coordinates:

Command: **line**

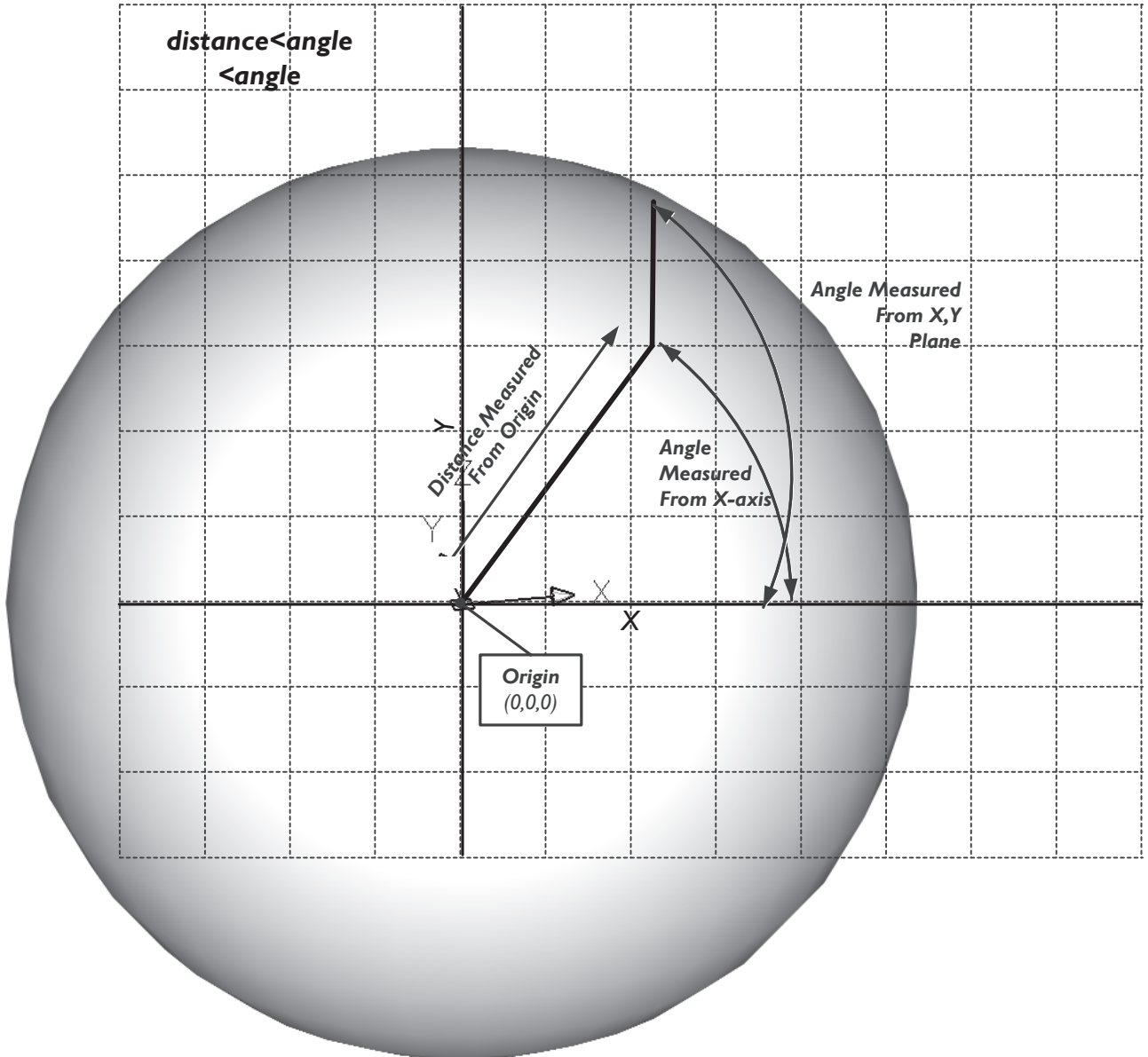
Specify first point: **1<23,4**

Specify next point or [Undo]: **5<67,8**

Spherical Coordinates

Spherical coordinates define 3D points by measuring a distance and two angles from the origin. The first angle lies in the x,y-plane, while the second angle is up (or down) from the x,y-plane. The distance lies, of necessity, in the x,y,z-plane.

To indicate the angles to AutoCAD, you prefix each with angle brackets, such as <15<45 for 15 and 45 degrees.



To draw a line using spherical coordinates:

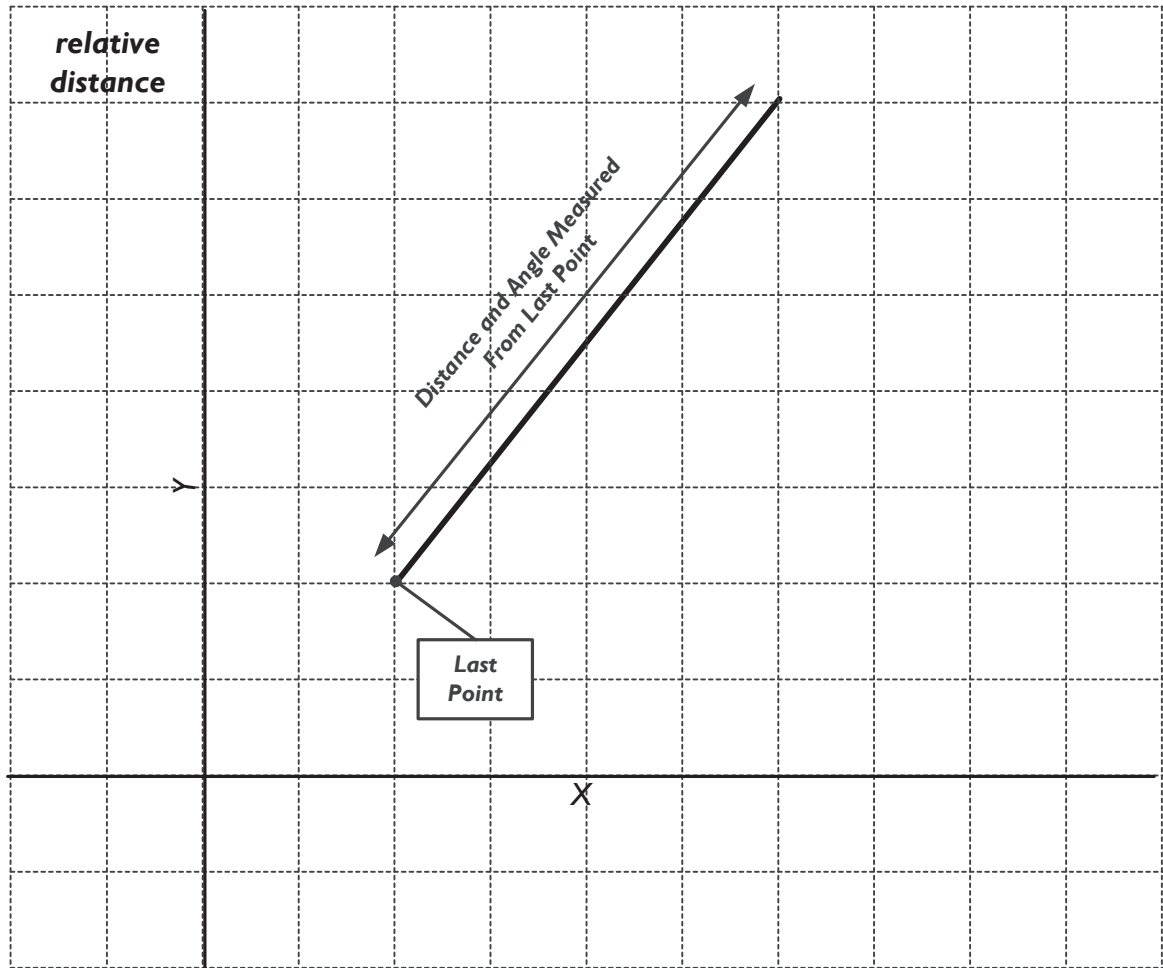
Command: **line**

Specify first point: **1<23<45**

Specify next point or [Undo]: **6<78<90**

Relative Coordinates

All coordinate systems can use *relative* coordinates, which measure the distance (and angle) from the last point. (The opposite of relative is *absolute*, which measures distances from the origin.) To indicate relative coordinates to AutoCAD, you prefix them with the at symbol (@), such as @2,3.



To draw a line using relative Cartesian, polar, cylindric, and spherical coordinates

Command: **line**

Specify first point: @1,2

Specify next point or [Undo]: @3,4,5

Specify next point or [Undo]: @6<78

Specify next point or [Undo]: @9<10<11

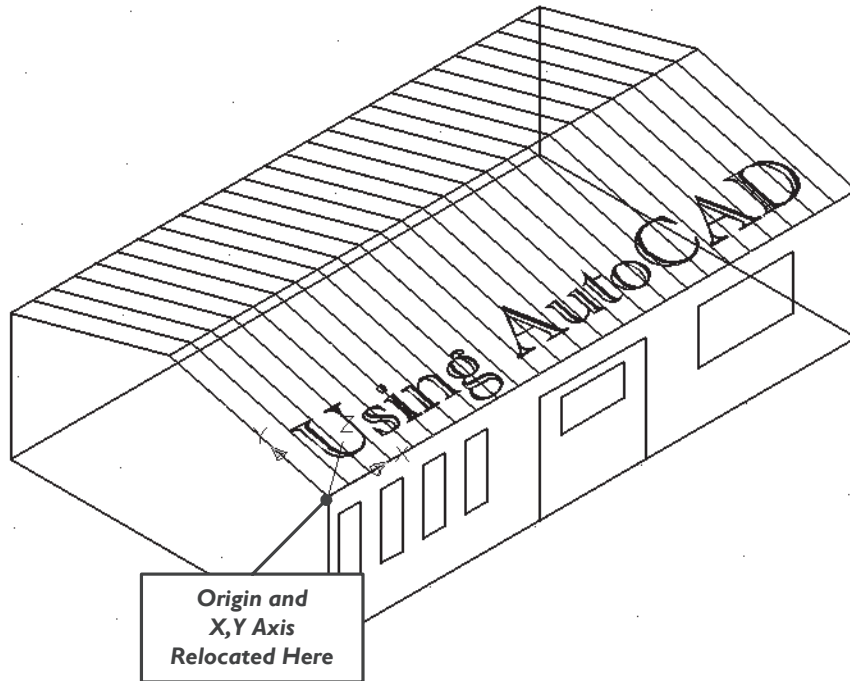
Specify next point or [Undo]: @9<10<11

Note that the first point can also be relative. It is measured from whatever last point was specified in the drawing with a previous drawing command. AutoCAD stores the coordinates of the last point in a system variable called **LASTPOINT**.

Similarly, relative angles are measured from the value stored in the LASTANGLE system variable.

User-defined Coordinate Systems

As I noted earlier, all drawing in AutoCAD takes place in the x,y-plane, also called the "working plane." How, then, do you draw in 3D space, which involves the z axis, along with the x,z and y,z planes? You could specify the z-coordinate, but this is not possible for all commands. In any case, specifying z coordinates becomes cumbersome when drafting details, say, on the side of a slanted roof, where the value of z is constantly changing.



AutoCAD takes care of that through the user-defined coordinate system, or "UCS" for short. Through UCSs, you can define the working plane anywhere in 3D space, such as on the sloped roof of a barn. (The UCS command lets you set the working plane relative to the current view, to three points picked in the drawing, and by several other methods.) UCSs can be saved by name, so that you can return to previously-created working planes.

ENTERING COORDINATES

Much of working with AutoCAD involves specifying points on the screen. Typical are the prompts of the LINE command:

Command: **line**

Specify first point: *(Pick a point, or enter x, y, z coordinates.)*

Specify next point or [Undo]: *(Pick another point, or enter more coordinates.)*

You can do this in three ways:

- Picking points on the screen.
- Entering coordinates at the keyboard.
- Using object snaps, tracking, direct distance entry, and point filters.

Screen Picks

When AutoCAD prompts you to specify a point, you can use the crosshair cursor to pick a point in the drawing. Move the mouse to move the cursor; press the left mouse button to pick the point. AutoCAD determines the x, y coordinates of the pick point.

Screen picks are strictly 2D (two dimensional), even when working in 3D (three dimensional) drawings. “2D” means AutoCAD records the x and y coordinates only, and assumes the z coordinate is 0. If you wish to specify a z coordinate, you may do this with the **ELEVATION** command (which sets the distance along the z axis), with the **.XY point filter** (which forces AutoCAD to prompt you for the z distance), or with an x, y, z triplet.

Keyboard Coordinates

You can enter coordinates at the keyboard in a variety of methods, as I described earlier. The formats permitted by AutoCAD are:

Absolute	Relative	Meaning
x,y	@x,y	2D Cartesian coordinates.
x,y,z	@x,y,z	3D Cartesian coordinates.
d<a	@d<a	2D polar coordinates.
d<a,z	@d<a,z	3D cylindrical coordinates.
d<a<r	@d<a<r	3D polar coordinates.

- x, y, and z = distance along the x, y, and z axis, respectively.
- d = distance in x,y plane.
- a = angle from x axis (in x,y plane).
- r = angle from x,y plane (in z direction).

Recall that *absolute* coordinates are measured relative to the origin (0,0,0), while *relative* coordinates are measured relative to the last point. Similarly, absolute angles are measured from the x-axis counterclockwise, while relative angles are measured from the value stored in the **LASTANGLE** system variable.

Snaps, Etc.

Entering coordinates at the keyboard can get tedious. Thus, there are alternatives and aids:

- **Point filters** combine screen picks with keyboard entry.
- **Snap spacing** restricts cursor movement to increments.
- **Polar snap** and **ortho mode** restrict cursor movement to specific angles.
- **Direct distance entry** allows you to show angles and enter distances.
- **Tracking** shows distances.
- **Object snaps** snap the cursor to the nearest geometric feature.
- **Object tracking** shows geometric relationships to nearby objects.

You learn more about snap mode, polar snaps, objects snaps, and object tracking in Chapter 5, “Drawing with Precision.”

Point Filters

Point filters combine screen picks with keyboard entry to specify, for example, the x coordinate with the keyboard and the y coordinate on the screen. AutoCAD recognizes these point filters:

Point Filter	AutoCAD Asks For...
.x	y and z coordinates.
.y	x and z coordinates.
.z	x and y coordinates.
.xy	z coordinate.
.xz	y coordinate.
.yz	x coordinate.

Here is an example of using point filters with the **LINE** command. Notice that you can enter coordinates at the keyboard or on the screen at any time:

Command: **line**

Specify first point: **.x**

of (*Pick a point; AutoCAD reads just the x-coordinate.*)

(need YZ): **2,3** (*AutoCAD reads these as the y and z coordinates.*)

Specify next point or [Undo]: **.y**

of **2** (*AutoCAD reads this as the y coordinate.*)

(need XZ): (*Pick a point; AutoCAD reads the x and z coordinates.*)

Snap Spacing

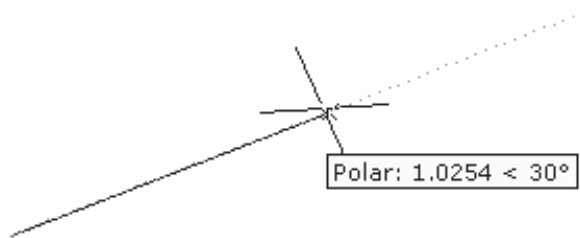
Snap mode restricts cursor movement. To restrict the cursor to moving in increments of 0.5, for example, set the snap spacing to 0.5 with the **SNAP** command. This allows you to pick points easily on the screen, accurate to the nearest 0.5 units. If you measured the rooms in your house to the nearest one inch, when drawing the floor plan in AutoCAD, you would set the snap spacing to 1".

Snap is often used in conjunction with the grid; when the grid is set to the same spacing as the snap, then you can “see” the snap points.

Polar Snap and Ortho Mode

Polar snap snaps the cursor to specific angles, while ortho mode restricts the cursor to 90-degree increments. (Ortho is short for “orthographic.”) You can specify the polar angle, although increments of 15 degrees are common. In addition, you can specify polar distances, such as 1.

While you are drawing, AutoCAD displays a tooltip showing relative distance and angle from the last point.



You turn on polar mode by clicking  (or **POLAR**) on the status bar; turn on ortho mode by clicking  (or **ORTHO**).

Direct Distance Entry

Direct distance entry shows angles, and types distances. It allows you to draw and edit by showing a relative distance: you move the cursor to indicate the angle, and then type a number to specify the distance. If you need a precise angle, then ensure that polar tracking or ortho mode is first turned on.

Specify first point: (*Move the cursor, and then enter a distance, such as 2.5.*)

Tracking

Tracking shows distances with the “pen up” during drawing and editing commands. You can think of it as the opposite of direct distance entry, which draws; tracking moves the cursor without drawing. During a prompt, such as “Specify first point:”, enter **tk** to invoke tracking mode. Move the cursor, in the direction to track, and then pick a point or enter a distance, such as **2.5**. You can continue

tracking as far as you need.

Specify first point: **tk**

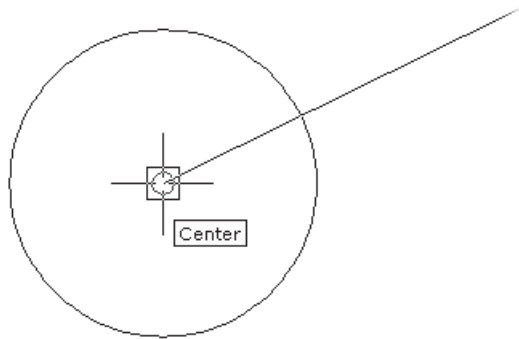
First tracking point: *(Move cursor, and then enter a distance or pick a point.)*

Next point (Press ENTER to end tracking): *(Press ENTER.)*

Object Snaps

Object snaps snap the cursor to the nearest geometric feature, such as the endpoints of lines, the center points of circles and arcs, and the insertion points of text. To help you find these geometric features, AutoCAD displays icons and tooltips.

The figure below illustrates the line being drawn to the center of the circle. Because object snapping is turned on, AutoCAD displays the center icon (a circle), as well as the tooltip, "Center."



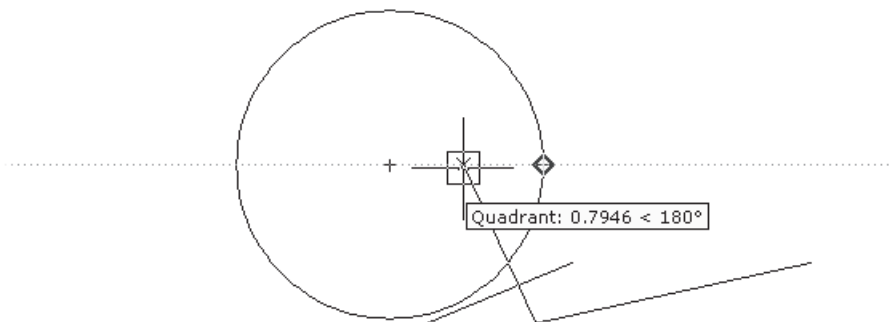
To use objects snaps, you can turn them on with the **OSNAP** command, or enter them at prompts, such as:

Specify first point: **center**

of *(Pick a circle or arc.)*

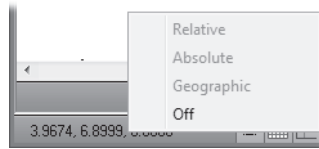
Object Tracking

Object tracking shows geometric relationships to nearby objects. When turned on with at least one object snap mode, object snap tracking shows *alignment paths* (dashed lines) based on object snap locations. The figure below illustrates the alignment path showing the quadrant object snap.



DISPLAYING COORDINATES

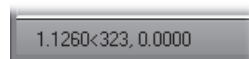
AutoCAD reports the cursor position by displaying the x, y, z coordinates on the status line (at the extreme left end, as illustrated below).



In most cases, the display shows absolute x, y, z coordinates; the z coordinate, however, always reads 0.0000 — unless changed by the ELEV command.

If you wish, you can turn off the display, although there is no point to doing that, at least not that I can think of. In addition, you can change the display to relative d<r coordinates. To change the coordinate display, press CTRL+I or function key F4 — or, right-click the coordinate display, and make the change in the popup menu.

In the figure below, notice that AutoCAD displays relative coordinates during line drawing:



1.1260 is the distance.

<323 is the angle

0.0000 is the z coordinate (elevation).

In addition to showing coordinates on the status line, AutoCAD also displays coordinate information through tooltips.

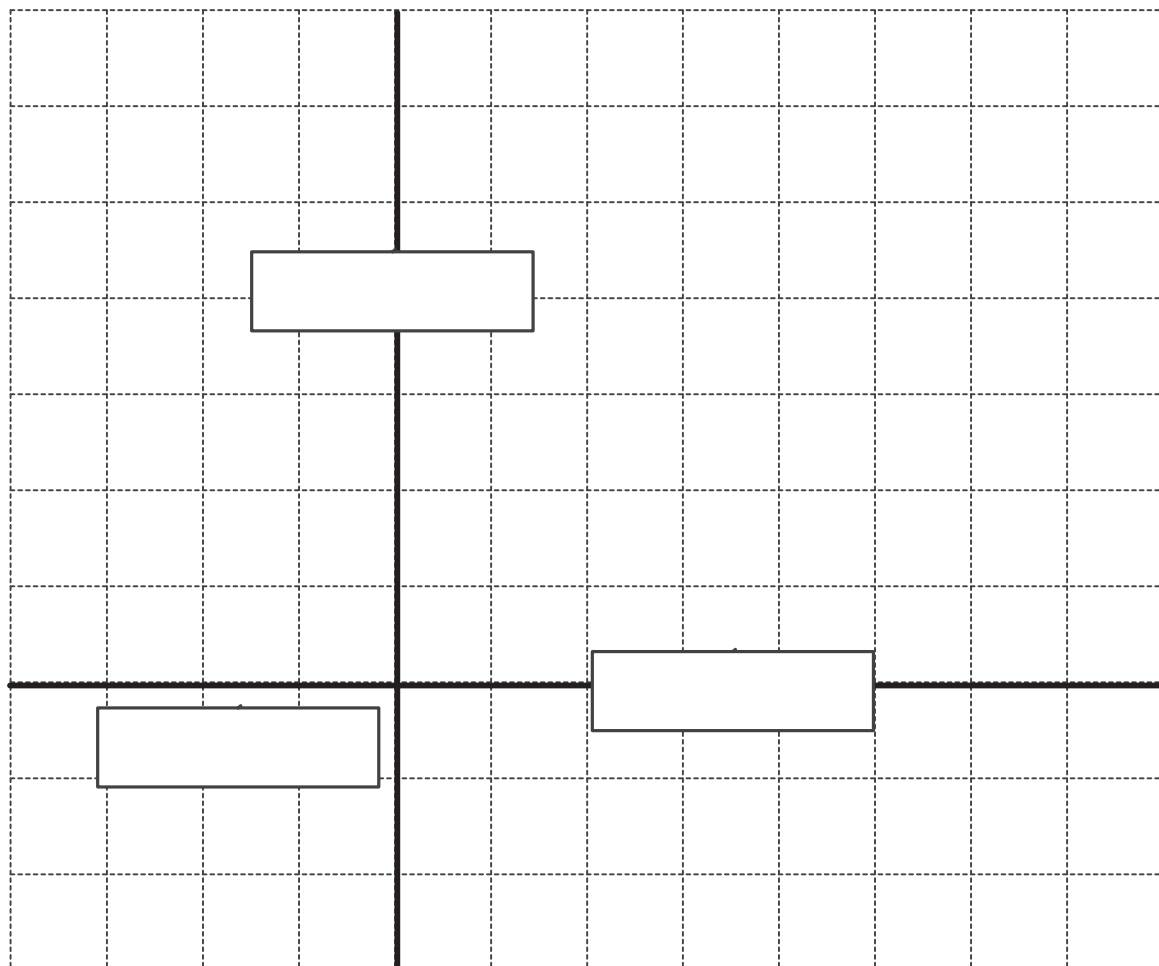
EXERCISES

1. In the following exercises, watch how the coordinates and cursor react to different modes.
Move the cursor around with your mouse. Do the coordinates at the lower left corner of the screen move?
Press **F6**. Move the cursor again, and watch the coordinates. What is the difference?
2. Press **F8**. Check the status bar: is ortho mode turned on?
Move the cursor. Do you notice a difference?
3. Press **F9** to turn on snap mode.
Move the cursor. Now is there a difference in cursor movement?
4. In this exercise, you practice using ribbon buttons and keystrokes that affect commands.
Move the crosshairs to the top of the drawing area and into the ribbon. Does the cursor change its shape?
In the Home tab, position the cursor over the arrow next to **Circle**, and then click.
Does a flyout drop down?
5. In the ribbon, choose **Line**.
Draw some lines.
To exit the command, right-click and select **Cancel** from the shortcut menu.
6. Press the **ENTER** key. Did the **Line** command repeat?
Clear the command line with **ESC**.
Press the spacebar. What happens?
7. Press **F2**. Do you see the Text window?
Press **F2** again to return to the drawing window.
8. Press **F1**. Do you see the Help window? Familiarize yourself with the Help window.
9. Use the **LINE** command to draw a line graph of insurance premiums for nonsmoking men. Enter the coordinates listed by the table below:

X (Age)	Y (Rate \$)
25	14
30	15
35	19
40	23
45	37
50	59
55	88
60	130
65	194

Are these coordinates relative or absolute? (If you cannot see the drawing, use the **Zoom Extents** command.)

10. Using the figure below, label each axis and the origin.



CHAPTER REVIEW

1. List two pieces of information found on AutoCAD's title bar.
 - a.
 - b.
2. What are the benefits of using *template* drawings?
3. Where are the mode indicators located?
4. How do you access command options during dynamic input?
5. Can you copy and delete files from within AutoCAD?
6. What happens when you press the **F2** key?
7. List the control and/or function key associated with the following actions:
 - Turns ortho mode on and off.
 - Selects all objects in the drawing.
 - Toggles between graphic and text screens.
 - Toggles tablet mode on and off.
 - Cancels all the characters on the command line.
 - Toggles to the next isometric plane in iso mode.
 - Toggles the screen coordinate display modes.
 - Toggles the grid on and off.
 - Toggles snap on and off.
8. After a command ends, what happens when you press **ENTER**?
9. Name ~~three~~ ways to invoke the on-line help facility:
 - a.
 - b.
 - c.
10. What is the purpose of the *prompt* area?
11. Which command turns the ribbon on?
 - Off?
12. What does the color BYLAYER mean?
13. What are *linetypes*?
14. Explain the purpose of the small square at the center of the crosshair cursor.
15. "UCS" is short for:
16. Can the UCS icon be turned off?
 - If so, why?
17. Explain the meaning of these numbers on the status line:
 - 32'-11 3/8", 1'-2 1/2", 0'-0"
18. What does it mean when the following buttons on the status line look bluish?
 - SNAP
 - OSNAP
 - GRID
19. Where is the *tray* located?
20. Can you "show" AutoCAD an angle by picking points in the drawing?
21. How do you show AutoCAD x and y displacement simultaneously?
22. Describe how *transparent* commands work.
23. Can any command be transparent?

24. What is an *alias*?
25. Write out the alias for each of the following commands:
 - ARC**
 - LINE**
 - BLOCK**
 - CIRCLE**
 - COPY**
 - ZOOM**
26. Where are *function keys* located?
How can you identify function keys?
27. Write out the function key you would use for the following actions:
 - Toggle snap mode
 - Toggle ortho mode
 - Toggle grid display
 - Exit AutoCAD
28. What does *toggle* mean?
29. Write out the **CTRL** key you would press for the following actions:
 - Display Properties palette
 - Toggle clean screen mode
 - Copy objects to the Clipboard
 - Save the drawing
 - Switch to the next drawing window
30. Describe two methods to repeat commands.
31. Explain in two or three words the function of each of these mouse buttons:
 - Left button
 - Right button
 - Scroll wheel
 - SHIFT**+Left button
32. How do you access the application menu?
33. Describe one way to display the menu bar.
34. List some pros and cons to using dialog boxes.
 - Pro:
 - Con:
35. Do all commands use dialog boxes?
36. Explain the difference between the **ARRAY** and the **-ARRAY** commands.
37. How do you force file-related commands to display their dialog boxes, when the function has been turned off?
38. Describe the purpose of the **Preview** button on dialog boxes.
39. What is a *non-modal* dialog box?
40. What is the purpose of *tabs* in dialog boxes?
41. Describe the difference between:
 - Check boxes
 - Radio buttons
42. What happens in dialog boxes when lists are too long?
43. How can you select more than one item from a list?

44. Describe the function of these commands:

LINE
CIRCLE
ARC
POLYGON
TEXT

Are these commands for drawing or editing?

45. Describe the function of these commands:

COPY
MIRROR
INSERT
MATCHPROP

46. Give two reasons why you might need to edit drawings?

47. Describe the function of these commands:

MOVE
ERASE
ROTATE
DDEDIT
PEDIT
PROPERTIES

48. What happens when you double-click a line of text?

Objects other than text?

49. Explain the difference between the two types of plotting:

Color-based.
 Styles-based.

50. Describe the function of *plot styles*.

51. List one way that the **PREVIEW** command is useful.

52. What is an *eplot*?

53. How would you email a drawing without sending the .dwg file itself?

54. What is “DWF” short for?

Name a benefit to .dwf files.

55. What happens if you forget the password to drawings?

56. Briefly explain these coordinate systems:

Cartesian
 Polar
 Spherical
 Cylindrical

57. Describe the meaning of the following symbols as used by AutoCAD:

<
 @
 , (comma)

58. Name the coordinate system used by the following:

25<45
 25,45,60
 25<45<60
 25<45,60

59. Write out relative coordinates for a distance of 5 units and an angle of 75 degrees.

60. Can coordinates be negative?
If yes, when?
61. What are the coordinates of the *origin*?
62. Are relative coordinates measured relative to the origin?
Are absolute coordinates measured relative to the origin?
63. In which direction are positive angles measured in AutoCAD, usually?
Can the direction be changed?
64. In which direction is 0 degrees in AutoCAD, usually?
Can the direction be changed?
65. Fill in the missing Cartesian coordinates to draw a line from the origin to 10,15.
Command: **line**
Specify first point:
Specify next point:
66. Fill in the missing relative coordinates to draw a vertical line 10 units long from 20,15.
Command: **line**
Specify first point:
Specify next point:
What are the coordinates of the line's endpoint?
67. What happens when you press **ENTER** at the **LINE** command's "Specify first point" prompt?
68. What is the likely z coordinate when you pick a point in the drawing?
When might the z coordinate change?
69. When might you use *cylindrical* and *spherical* coordinates?
70. When might you use UCSs?
71. Describe the distances and angles employed by spherical coordinates:
Distance
Angle 1
Angle 2
Write down an example of a spherical coordinate.
72. How are *point filters* helpful?
73. When you enter **.xy**, what does AutoCAD ask for?
74. Explain the purpose of:
Snap spacing
Object snaps
Polar snaps.
75. What is the difference between *direct distance entry* and *tracking*?
76. How do you enter tracking mode?
77. Can angles be negative?
If so, when?
78. Describe what happens when you press the **Esc** key at the 'Command:' prompt.
79. List five ways to enter commands:
 - a.
 - b.
 - c.
 - d.
 - e.

80. What happens when you enter the first few letters of a command name, and then press the **TAB** key?
81. What is *dynamic input*?
82. How is dynamic input activated?
83. Can the x, y coordinates be changed during dynamic input?
If so, how?
84. What is the meaning of the dot (•) next to an option in dynamic input?
85. From where are dynamic angles measured?
86. From where are dynamic dimensions measured?

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Setting Up Drawings

After starting a new drawing, you must prepare it for use. This chapter covers the following items to help you set up drawings:

NEW starts new drawings from scratch, from a template, or through the Startup dialog box.

QNEW starts new drawings based on templates.

Advanced Wizard sets up units and angles.

OPEN opens existing drawings, and helps manage files.

OPTIONS changes user interface colors, specifies backup options, and much more.

UNITS specifies unit and angle formats.

Scale factors affect the size of text, linetypes, hatch patterns, and dimensions.

SAVE and **SAVEAS** save drawings by other names, in other formats, and to other computers.

QSAVE saves drawings quickly.

QUIT exits AutoCAD.

NEW TO AUTOCAD 2010 IN THIS CHAPTER

- The menu browser is now known as the “application menu.”

NEW DRAWINGS

When you start AutoCAD, you either begin with a new drawing or call up previously-saved drawings. Let's begin with starting a new drawing.

After AutoCAD starts, it displays a new blank drawing. (At the same time, it may also display a variety of startup dialog boxes. If the New Features Workshop dialog box appears, cancel it.)

You can use the **NEW** or **QNEW** commands at any time to start additional new drawings, as described below. You can have many drawings open in AutoCAD at the same time.

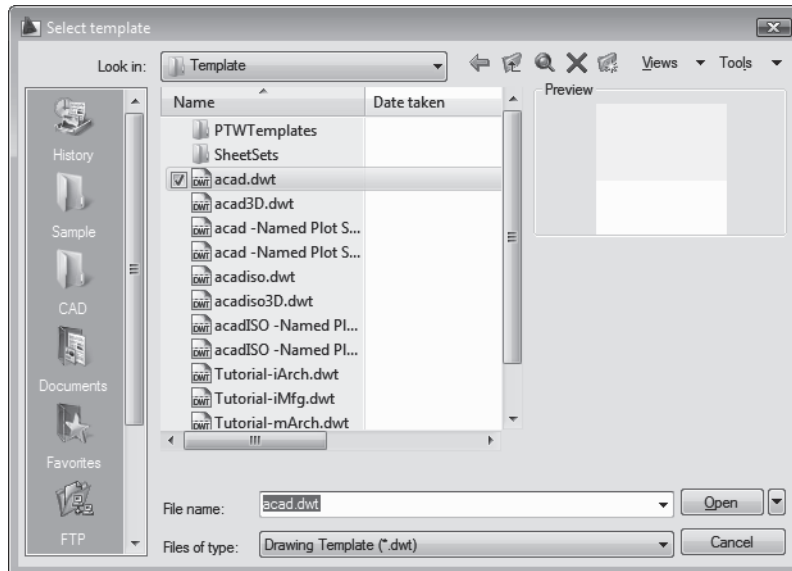
TUTORIAL: STARTING NEW DRAWINGS

- To open a new drawing when AutoCAD is already running, start the **NEW** command by one of the following methods:
 - In the application menu, choose **New**.
 - Alternatively, press the **CTRL+N** keyboard shortcut.
 - Or, at the 'Command:' prompt, enter the **new** command.

Command: **new** (Press ENTER.)

- In some cases, AutoCAD may display the Startup dialog box. But usually, AutoCAD prompts you to select a template file.

Select a template file, which determines what new drawings look like. Some are blank, while others contain drawing borders and other elements. If you are unsure, select *acad.dwt*.



- Click **Open**. Notice that AutoCAD displays a new drawing; if based on *acad.dwt*, the new drawing is blank.

About Templates

Each time you begin a new drawing, AutoCAD makes an exact copy of a *template* drawing. (These drawing files are identified with the extension of *.dwt*.) AutoCAD copies the settings from template files, which have names such as *acad.dwt*, *acadiso.dwt*, and *acad3d.dwt*.

Template drawings contain settings that new drawings use; sometimes, templates also include drawing elements. New drawings are identical to the template, except for the file name. You can change any setting in the drawings, regardless of how it is defined in the template.

AutoCAD stores its list of template drawings in a folder with the long path of */documents and settings/<user login name>/local settings/application data/autodesk/autocad 2010/r18.0/enu/template*. AutoCAD includes a number of template drawings for a variety of sizes and standards.

As you select a file name, AutoCAD displays its preview image. Clicking **Open** opens the template, and then this file becomes the starting point for your new drawing.



The **QNEW** command is a more flexible version of **NEW**: depending on how AutoCAD is set up, the command either starts with a preselected *.dwt* file, or else prompts you to select the template. (**QNEW** is short for “quick new.”)

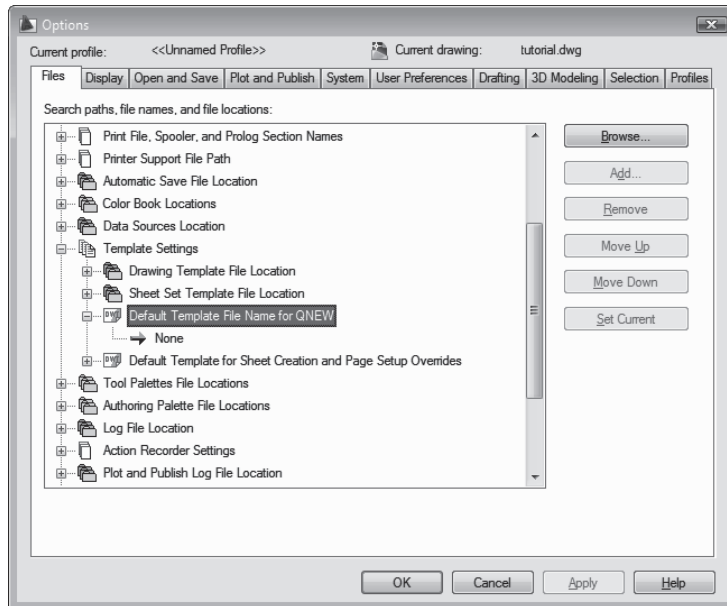


Notes Whether AutoCAD displays the Select Template dialog box is determined by a setting in the Options dialog box. From the **Tools** menu, select **Options**, choose the **Files** tab, and then look for the Template Settings section.

The **Default Template File Name for QNEW** item has two options:

- If **None** is shown, then the **QNEW** command displays the Select Template dialog box, just like the **NEW** command.
- If a file name with a *.dwt* extension is shown, then the **QNEW** command starts new drawings with the specified *.dwt* template file. This is the “quick” version of the command.

To select a specific template or another drawing, click the **Browse** button, and then choose the file.



TUTORIAL: STARTING NEW DRAWINGS WITH TEMPLATES

To open a new drawing based on a template, start the **QNEW** command:

- On the Quick Access toolbar, click the **New** button.
- Or, at the ‘Command:’ prompt, enter the **qnew** command.

Command: **qnew** (Press ENTER.)

If a template file is defined as described above, then AutoCAD displays a new, blank drawing based on that preselected *.dwt* file.

STARTUP DIALOG BOX

If AutoCAD displays the Startup dialog box, then you are faced with the following options:

Start from Scratch starts new drawings in metric or English units.

Use a Template starts new drawings based on .dwt template files.

Use a Wizard starts new drawings based on two wizards, Quick and Advanced.

Open a Drawing opens existing drawings.

(The dialog box discussed here is displayed when AutoCAD first starts; the nearly-identical Create New Drawing dialog box is displayed when you enter the NEW command. The only difference between the two, other than the title, is that the Create New Drawing dialog box grays out the **Open a Drawing** button, because this function is handled by the OPEN command.)



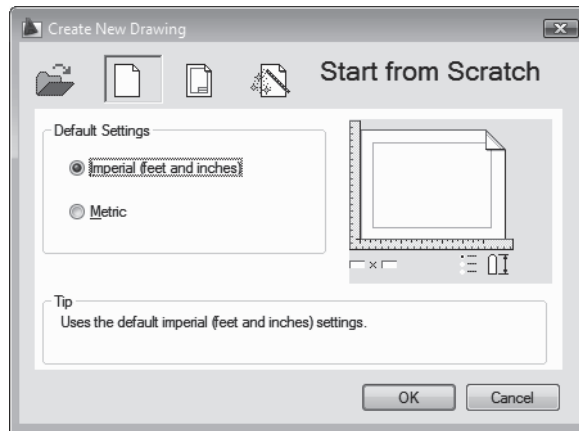
Note Whether AutoCAD displays the Create New Drawing dialog box is determined by the value of the **STARTUP** system variable:

1 — displays the Startup dialog box.

0 — does not display the dialog box (default setting).

Starting From Scratch

The **Start from Scratch** option is one way to start new drawings. It presents just two options: measuring the drawing in either Imperial (English) or metric units.



Select **Imperial** to use feet and inches. The drawing limits are set to 12" x 9", and dimension styles to inches. You can change these settings later. (Whether AutoCAD copies the settings stored in the *acad.dwt* template file seems to change from release to release; in 2010, it does not).

Select **Metric** for metric units. Limits are set to 429 mm x 297 mm, and the dimension styles to metric. AutoCAD copies the settings stored in the *acadiso.dwt* template file.

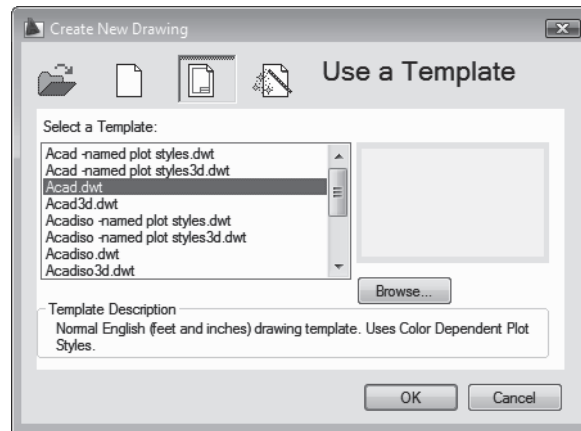
Behind the scenes, AutoCAD stores your selection in the **MEASUREUNIT** system variable, which has a value of 0 or 1:

0 — (English) AutoCAD reads the hatch patterns and linetypes defined by the **ANSIHatch** and **ANSILinetype** settings in the Windows registry.

1 — (Metric) AutoCAD reads the hatch patterns and linetypes defined by the **ISOHatch** and **ISOLinetype** registry settings.

Using Templates

In the Setup dialog box, choose the **Use a Template** button. AutoCAD displays the same list of template drawings as it did with the NEW command.

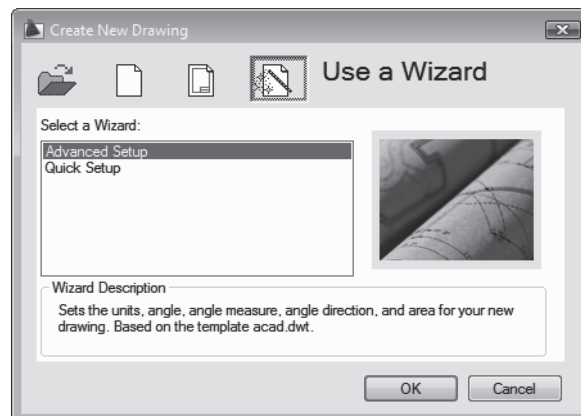


Select a file name, and AutoCAD displays its preview image. Click **Open** to open the template, which is the starting point for your new drawing.

Although template drawings have the *.dwt* extension, you are free to use *any* AutoCAD drawing as a template. Choose **Browse** to find other files. Or, in the Select a Template dialog box, click the **Files of type** droplist, and select *.dwg*.

Using Wizards

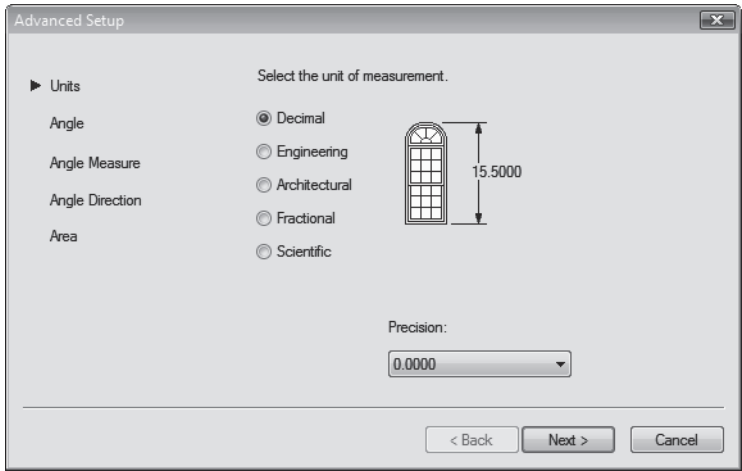
AutoCAD includes two “wizards” that step you through the stages of setting up some of the many parameters that define a drawing: Quick and Advanced.



The Quick Wizard takes you through just two steps — units and area — in creating new drawings. You previously worked with the Quick Wizard in Chapter 1, “Quick Start in AutoCAD.” Below, you work with the Advanced Wizard, which takes you through five steps to set up units, angles, and area for new drawings.

TUTORIAL: USING THE ADVANCED WIZARD

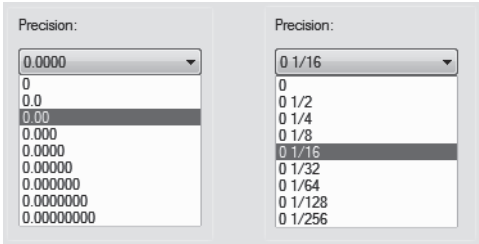
1. To start a new drawing with the advanced wizard, choose the **Use a Wizard** button.
2. Select **Advanced Wizard**.
Advanced Setup presents a series of dialog boxes that leads you through the steps of setting up new drawings. Curiously, no online help is available for this early encounter with AutoCAD.
3. The Units dialog box asks you to specify the units.
Select a measurement unit. If you are not sure, select **Decimal**. (If you change your mind later, you can change the units with the **UNITS** command.)



Technically, AutoCAD does not work with Imperial or metric units; instead, the software uses scientific units internally. To help human operators, however, AutoCAD accepts and displays units in the formats listed below, converting to and from scientific format on the fly.

<i>AutoCAD Units</i>	<i>Example</i>	<i>Comments</i>
Decimal	14.5	Decimal and metric units.
Engineering	1'-2.5"	Feet and decimal inches.
Architectural	1'-2 1/2"	Feet and fractional inches.
Fractional	14 1/2	Fractional inches, no feet.
Scientific	1.45E+01	Exponent notation.

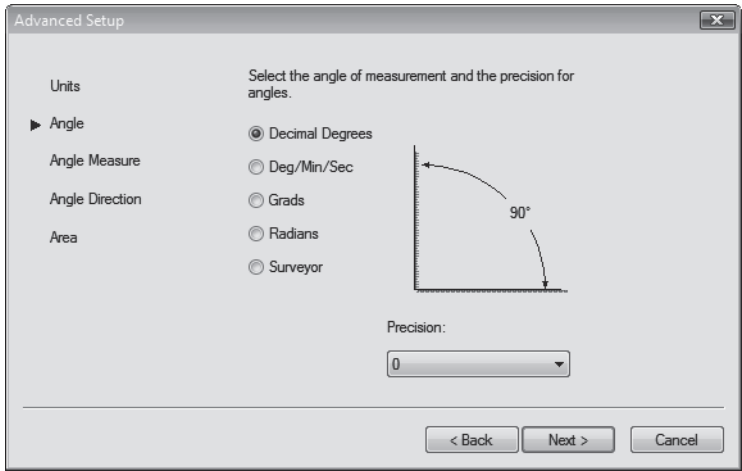
4. *Precision* refers to the number of decimal places or fractions of an inch. From the **Precision** droplist, select the number of decimal places (or fractional accuracy). If you are not sure, select 2 decimal places (0.00) or 1/8 fractional.



Left: Selecting decimal precision.
Right: Selecting fractional precision.

This affects the display of units only; no matter which precision you select, AutoCAD calculates to 16 digits internally, and then displays the result rounded to the maximum of eight places or 1/256th.

- Decimal, engineering, and scientific units: 0 to 8 decimal places.
 - Architectural and fractional units: 1/1, 1/2, 1/4, 1/8, 1/16, 1/32, 1/64, 1/128, or 1/256.
5. Click **Next**.
6. The Angle dialog box specifies the style of angle measurement. Select one of them. If you are not sure, select decimal degrees.



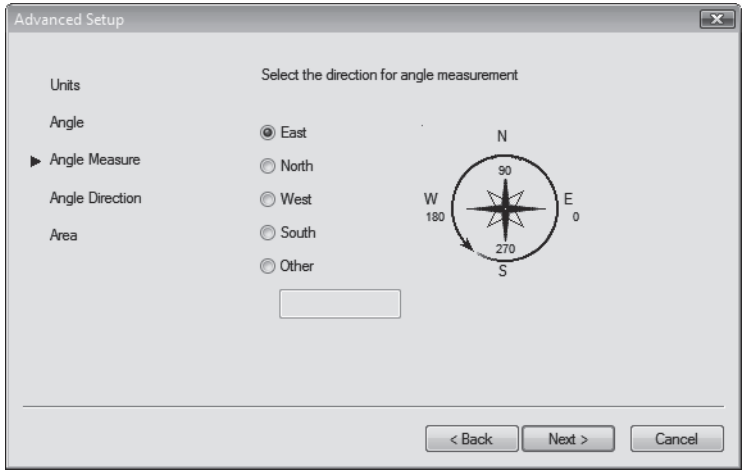
AutoCAD works internally with radians, but accepts and displays angles in the following formats:

Angle Formats	Example	Comments
Decimal Degrees	45.00	360 degrees in a circle.
Degrees-minutes-seconds	45d00'00.00"	Fractions of seconds in decimals.
Grads	50.00g	400 grads in a circle.
Radians	0.79r	2pi radian in a circle.
Surveyor	N45d0'0.00"E	Fractions of seconds in decimals.

7. As with units, Precision affects only the *display* of angles:
- Decimal degrees, grads, and radians: 0 to 8 decimal places.
 - Deg/min/sec and surveyor angles: 45d, 45d00' through to 45d00'00.00000".

From the **Precision** droplist, select the number of decimal places. (There is no fractional accuracy for angles.) If you are not sure, select 1 decimal place.

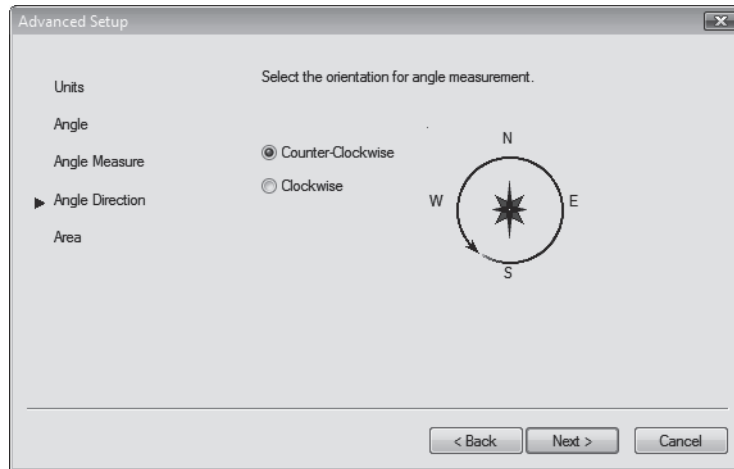
8. Click **Next**.
9. The Angle Measure dialog box asks you to “Select the direction for angle measurement.” The dialog box would be more accurate if it read, “Select the direction for 0 degrees.”



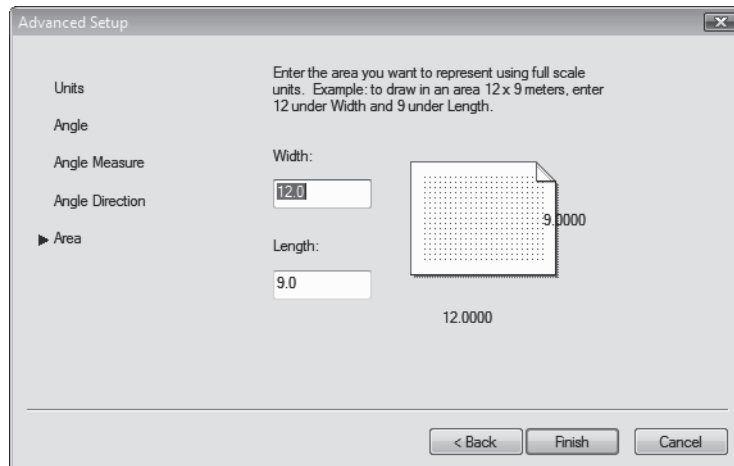
The default is **East**, which is the correct setting for most drawings. Measuring angles from East is the same as measuring from the positive x axis.

If your drawings use a direction other than 0 degrees, select from **North**, **West**, **South**, or **Other**. In the Other text box, you enter an angle in degrees between 0 and 360 — even when you selected grads or radians in the previous dialog box.

10. Click **Next**.
11. The Angle Direction dialog box asks you for the direction in which to measure positive angles — counterclockwise or clockwise. For most drawings, you keep the direction counterclockwise.



12. Click **Next**.
13. The Area dialog box asks you to specify the limits of the drawing. The default is 12 units wide (x direction) by 9 units tall (long, y direction).



If you know how large the drawing will be, enter values in the **Width** and **Length** text boxes. Clear the current values using the **DEL** or **BACKSPACE** key.

AutoCAD uses these values to set the limits of the drawing, which affect the extent of the 2D grid display (as shown in the preview image above) and the **Zoom All** command. You can always change the limits later with the **LIMITS** command.

(History: AutoCAD prevented you from drawing outside the area defined by the limits; this didn't make much sense, and Autodesk soon dropped the restriction with 1986's version 2.17g, adding the **LIMCHECK** system variable.)

14. Click **Finish**.
AutoCAD opens the new drawing with the settings you specified.

OPEN

The **OPEN** command opens existing drawings.

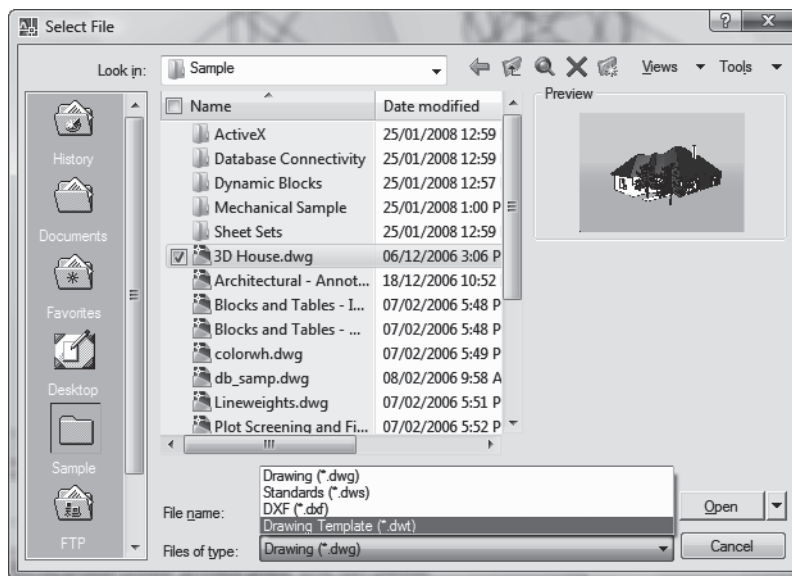
This command opens drawings for editing. It opens files previously saved in only AutoCAD's formats; other commands in AutoCAD open files in non-AutoCAD formats.

TUTORIAL: OPENING DRAWINGS

- I. To open drawings, start the **OPEN** command:
 - On the Quick Access toolbar, click the **Open** button.
 - From application menu, choose **Open**.
 - Alternatively, press the **CTRL+O** keyboard shortcut.
 - Or, at the 'Command:' prompt, enter the **open** command.

Command: **open** (Press ENTER.)

In most cases, AutoCAD displays the Select File dialog box.



2. Select a **.dwg** file from the list, and then choose **Open**.
Notice that AutoCAD opens the drawing.

The Select File dialog box accesses the following functions:

- Opens **.dwg** (drawing), **.dxf** (interchange), **.dws** (standards) and **.dwt** (template) files.
- Selects one or more files to open at the same time.
- Previews drawings before opening, via thumbnail representation.
- Sorts files by name, date, type, and size.
- Views names of drawings recently opened (History), and most often used (Favorites).
- Opens drawings protected against change (their status set to "read-only").
- Loads all or portions of drawings.
- Accesses drawings stored on your computer, on any other computer located on your network, or from any location accessible through the Internet (FTP).

- Searches for drawings anywhere on your computer (Find).
- Searches the Internet for information.
- Creates a list of frequently-accessed folders.
- Creates new folders and manipulates files and folders, including renaming and deleting files.
- Sends drawings as email messages, or to removable drives.
- Resizes the Select Drawing dialog box.
- Accesses Autodesk's Buzzsaw Web site for managing construction projects.

Moving counterclockwise, the Select File dialog box consists of five primary sections:

Files List lists the names of files and folders.

Preview displays a thumbnail-size image of the drawing.

Files of Type specifies the name and type of file to be opened.

Standard Folders Sidebar provides shortcuts to folders and Web locations.

Toolbar holds tools useful for manipulating files and folders.



Note While the **OPEN** command is the most common method to open files, it is not the only way:

INSERT inserts other drawings as blocks into the current drawing.

PARTIALLOAD loads additional portions of partially-loaded drawings.

FILEOPEN opens drawings without a dialog box; useful for scripts and macros, but is limited to AutoCAD running in SDI (single drawing interface) mode.

ATTACH attaches images and files in DWF, DWFX, DGN, and PDF formats as underlays.

OPENDWFMARKUP attaches marked-up .dwf files.

OPENSHEETSET opens .dst sheet set data files and related drawings.

DXBIN opens .dxb (drawing exchange binary) files created by CAD\camera (obsolete).

DXFIN opens .dxf (drawing interchange format) files created by AutoCAD and other CAD programs.

ACISIN imports .sat format solid models created by ACIS-compatible CAD programs.

WMFIN inserts .wmf (Windows meta format) files.

XOPEN opens externally-referenced drawings in their own windows.

3DSIN opens .3ds models created by 3D Studio.

And the Clipboard pastes vector and raster graphics and text into drawings from almost any program.

Files List

To open drawings:

1. Select the name from the Files list. Notice that the name appears in the **File name** field.
2. Click **Open**.

As a shortcut, you can double-click the file name. This is equivalent to selecting the file name, and then clicking the **Open** button.

Preview

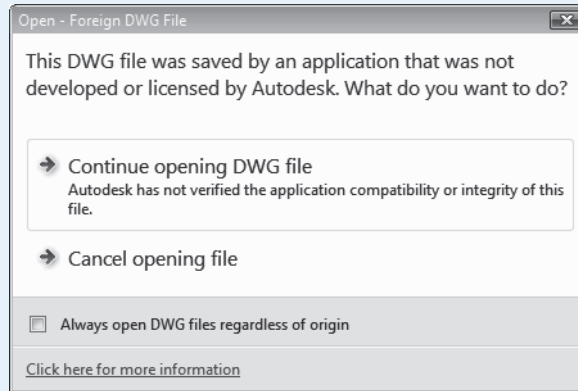
When you select a drawing in the Files list, AutoCAD displays its preview image. The image is small, but provides sufficient detail to distinguish it from most other drawings. The preview image, also called a “thumbnail,” is generated automatically by AutoCAD when you save the drawing. Because the thumbnail shows the view when the drawing was last saved, the view may not include the entire drawing.

TRUSTED DWG & DWGCHECK

When AutoCAD opens a drawing, it reports on its source. When the drawing was last saved by Autodesk software, the following prompt appears in the command bar:

Autodesk DWG. This file is a TrustedDWG last saved by an Autodesk application or Autodesk licensed application.

When the drawing comes from a non-Autodesk source, a warning appears in a dialog box:



- **Continue** opens the drawing, and then repeats the warning at the command line:
Non Autodesk DWG. This DWG file was saved by a software application that was not developed or licensed by Autodesk. Autodesk cannot guarantee the application compatibility or integrity of this file.
- **Cancel** does not open the drawing.
- **For more information click here** opens Internet Explorer with an Autodesk Web page that explains Trusted DWG (www.autodesk.com/trusteddwg).
- ☐ **Do not show me this again** stops the dialog box from appearing in the future.

AutoCAD 2007 introduced *Trusted DWG*, because Autodesk felt that drawings edited and saved by non-Autodesk software may corrupt data and destabilize applications used with AutoCAD. Trusted DWG does not repair defects in files, but allows you to choose whether to load these “foreign” .dwg files.

The stern wording of the dialog box was changed in AutoCAD 2008 after Autodesk launched a law suit against the Open Design Alliance. The alliance had used the registered term “Trusted DWG.” As part of the settlement, the alliance agreed to stop using the term, and Autodesk agreed to tone down the wording of the warning.

DWGCHECK

The **DWGCHECK** system variable determines how and when these warnings are displayed.

- 0** — When the drawing has a possible problem, the warning appears in a dialog box.
- 1** — When the drawing was saved by software other than AutoCAD or AutoCAD LT or when it has a possible problem, the warning appears in a dialog box.
- 2** — When the drawing has a possible problem, the warning appears on the command line.
- 3** — When the drawing was saved by software other than AutoCAD or AutoCAD LT or when it has a possible problem, the warning appears on the command line.

If you do not want the warnings to appear, turn off **DWGCHECK** by setting its value to 0, as follows:

Command: **dwgcheck**

Enter new value for DWGCHECK <1>: **0**

(History: The first several releases of AutoCAD presented a numbered *menu* of choices. To open a drawing, users pressed **2**, and then ENTER. Option 2 was labeled “Edit an EXISTING drawing.”

AutoCAD then prompted for the name of the drawing; users had to memorize the drawing's file name, because they could not select file names from a list. Release 9 introduced the basic dialog box that allowed users to select drawing names, as well as to choose the drive and folder names. AutoCAD 2000i greatly expanded the options through the dialog box you see today.)



Notes If you see no preview image, the **RASTERPREVIEW** system variable was turned off (set to 0) when the drawing was saved. Turn it on by changing the value to 1:

Command: **rasterpreview**

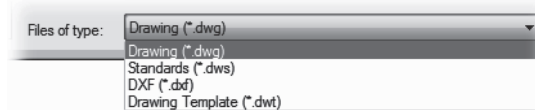
Enter new value for RASTERPREVIEW <0>: **1** (Press ENTER.)

The other possibility is that the drawing files were created by old versions of AutoCAD or other types of CAD software that don't include preview images.

Drawings in 3D are shown rendered when the drawing was saved with a visual style turned on.

Files of Type

By default, the dialog box displays the names of *.dwg* files, which are AutoCAD drawings. In addition, you can select other kinds of files to open:



To select a file type other than from *.dwg*, click the **Files of type** list:

DWG — (drawing) for creating AutoCAD drawing files.

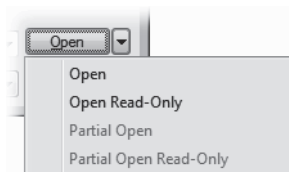
DWS — (drawing standards) for ensuring drawings use standardized elements, such as linetypes and layers, against which drawings are checked.

DXF — (drawing interchange format) for creating drawings that can be exchanged between otherwise incompatible programs.

DWT — (drawing template) for creating preset frameworks of new drawings, such as dimension styles and drawing borders. When you select "Drawing Template (*.DWT)", the dialog box automatically switches to the *\template* folder.

If you are not sure, select *.dwg*.

There is more than one way to open a drawing file. Click the down-arrow next to the **Open** button.



Read-Only

Notice the menu that appears:

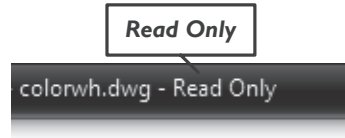
Open — loads all of the drawing; you can edit the drawing, and save the changes.

Open Read-Only — loads all of the drawing, and does not allow you to save changes unless saved by a different file name.

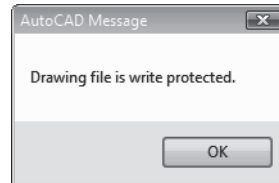
Partial Open — loads parts of the drawing, based on layer and view names.

Partial Open Read-Only — loads parts of the drawing, and does not save editing changes.

When you select one of the read-only options, AutoCAD displays the “Read Only” message on the title bar, as illustrated below.

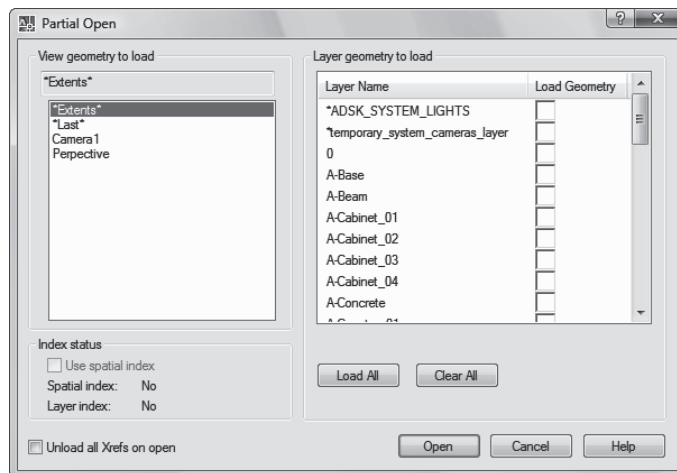


If you want to save read-only drawings, the Save As dialog box appears. When you attempt to save the drawing with the same name, the warning illustrated below is displayed. This prevents you from saving the drawing to the original file. (Give it a different file name, as described later in this chapter.)



Partial Open

When you select one of the partial-open options, AutoCAD displays the Partial Open dialog box. (*Partial-open* means that only part of the drawing is loaded, useful for very large drawings.)



Select a view from the left hand column and the layers to be opened in the right hand column. When done, click **Open**. The partial opening of drawings is described in detail in *Using AutoCAD: Advanced*.

Select Initial View

Getting back to the Select File dialog box, the **Select Initial View** check box determines whether the drawing is opened with a saved view:



- ☐ (Off) When the drawing is opened, AutoCAD displays the same view as when the drawing was last saved.
- ☒ (On) Before the drawing is opened, AutoCAD displays the Select Initial View dialog box, which lets you select a named view.

If no views have been saved, then the only option is *Last View*, the view of the drawing when it was last saved. Views are described in Chapter 5, “Drawing with Precision.”

Making Dialog Boxes Bigger

When you first see the dialog box, it is at its smallest. You can make it larger to see more file names at a time. With the cursor, grab an edge of the dialog box, and stretch the dialog box larger, revealing more file names.



Note Press function key **F5** to update the files list. You may need to do this when files appear to be missing or incorrectly named.

Standard Folders Sidebar

A vertical list of folder names, called the “sidebar,” is located on the left of the dialog box. It directly accesses folders on your computer, networks, and the Internet. From top to bottom, these are:

History displays a list of the most-recently opened drawings.

My Documents displays files stored in your computer’s `\my documents` folder.

Favorites displays files stored in the `\windows\favorites` folder; add files to this folder with **Tools | Add to Favorites**.

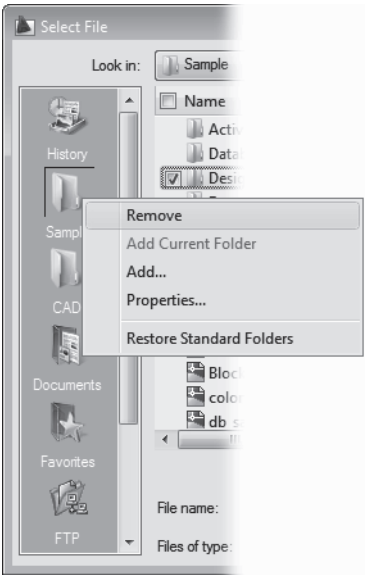
FTP browses FTP (file transfer protocol) sites on the Internet; add more sites with **Tools | Add/Modify FTP Locations**.

Desktop displays the files and folders found on the Windows “desktop.”

Buzzsaw connects you to Autodesk’s www.buzzsaw.com Web site, after you install their ProjectPoint client software.

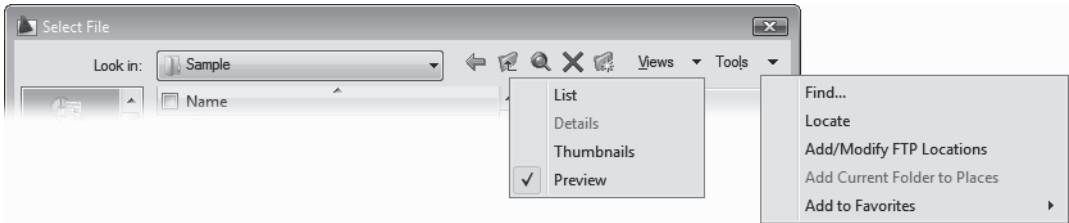
To add folders to the list, simply drag them from the Files list. For example, I like to have AutoCAD’s `\sample` folder on the list for fast access to the sample drawings, so I drag it onto the sidebar.

Commands for editing the sidebar are available by right-clicking the folder sidebar, as illustrated by the figure.



Finding Files

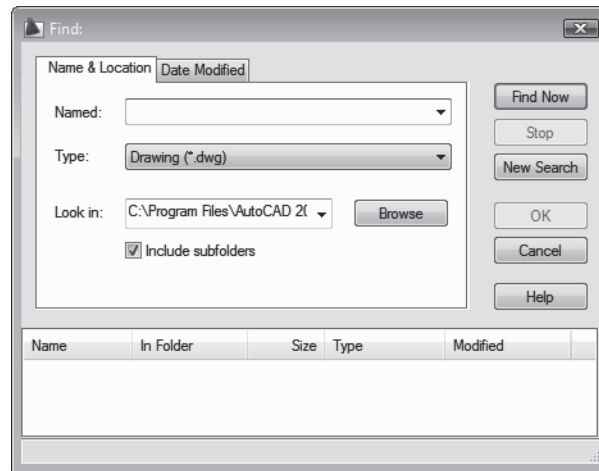
Sometimes you cannot remember the names of drawing files or where they are located. To help you locate them, AutoCAD includes the **Find** option in the **Tools** item on the dialog box’s toolbar.



When you choose **Find**, AutoCAD displays a Windows dialog box for finding files.

It is able to look at every folder on every drive in your computer. Drives include the hard drives, external drives, CD/DVD drives, and other storage devices. In addition, if your computer is connected to a network of other computers, the Find feature can search every folder of every networked drive your computer has permission to read.

The **Named** text box allows you to enter as much of the name as you can remember. Use wildcard characters to specify part of a name.



Recall that `?` is a placeholder for any character, while `*` searches for all files that match the rest of the criteria you supply. For example, `door*` searches for all files that start with **Door**:

door door36 doorstop

The **Type** droplist provides a filter for selecting `.dwg` drawing, interchange `.dxf`, template `.dwt`, and standards `.dws` files.

The **Look In** droplist restricts the search to specific drives, folders, and paths.

The **Date Modified** item narrows the search to specific dates. This is useful when you know roughly the date the drawing was created. For example, if the drawing was created last month, specify “Between these dates,” and then enter the month.

After you enter the search parameters, choose **Find Now**. Windows spends a bit of time rummaging through your computer’s drives, and produces a list of matching files. At any time, choose **Stop** to bring the process to a premature halt.

CHANGING SCREEN COLORS

When first installed on your computer, AutoCAD is designed to start up in a workspace called “2D Drafting & Annotation.” Other workspace names include AutoCAD Classic and 3D Modeling:

- **2D Drafting & Annotation** is the default workspace for AutoCAD 2010.
- **3D Modeling** sets up interface elements for 3D modeling.
- **AutoCAD Classic** makes AutoCAD look like it did before 2009.

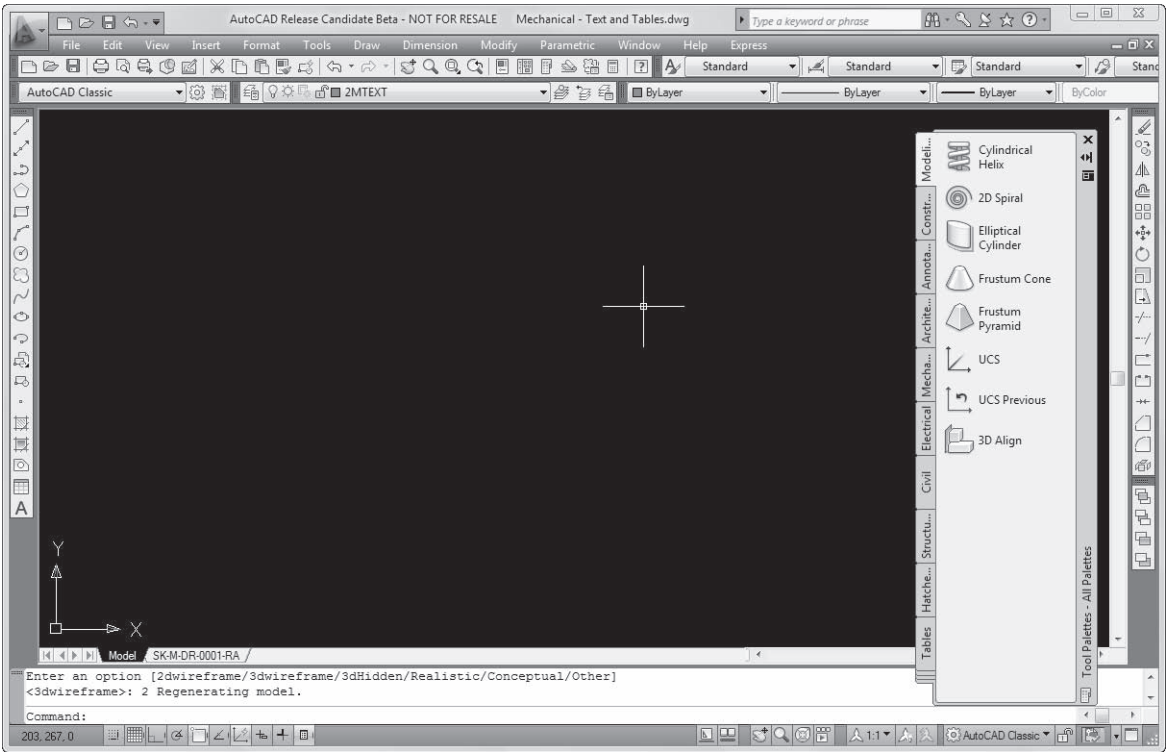
You can change workspaces through the Workspaces button on the status bar.

Default Background Colors

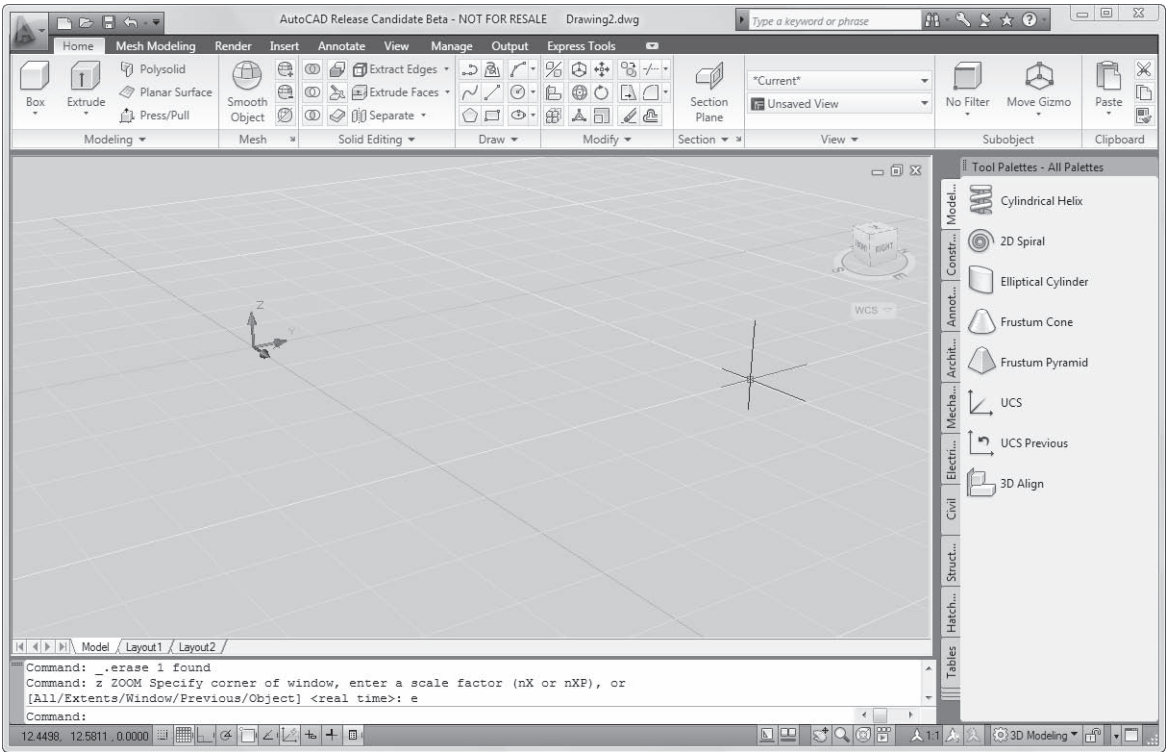
In the 2D workspaces, AutoCAD displays the drawing area of model space in pale yellow. Some drafters prefer black or white as the background color. Black makes the colors vibrant, while white simulates paper.

(History: In the past, computer screens had black backgrounds, displaying the text and graphics in monochrome colors, such as green, amber, or white — hence the name, “monochrome display.” With the advent of Macintosh and its paper metaphor, the background color of choice became white, because it is the color of paper and other media upon which drawings are printed. The white background was then mimicked by Windows. Still, some drafters prefer the black background — the default background color until AutoCAD 2008 — because it makes lighter colors, such as yellow and green, stand out better; others like it because black can create less eyestrain. And, added the technical editor, black grid dots are nearly invisible on the white background.)

Below, I've changed the background color of the “AutoCAD Classic” workspace to black.



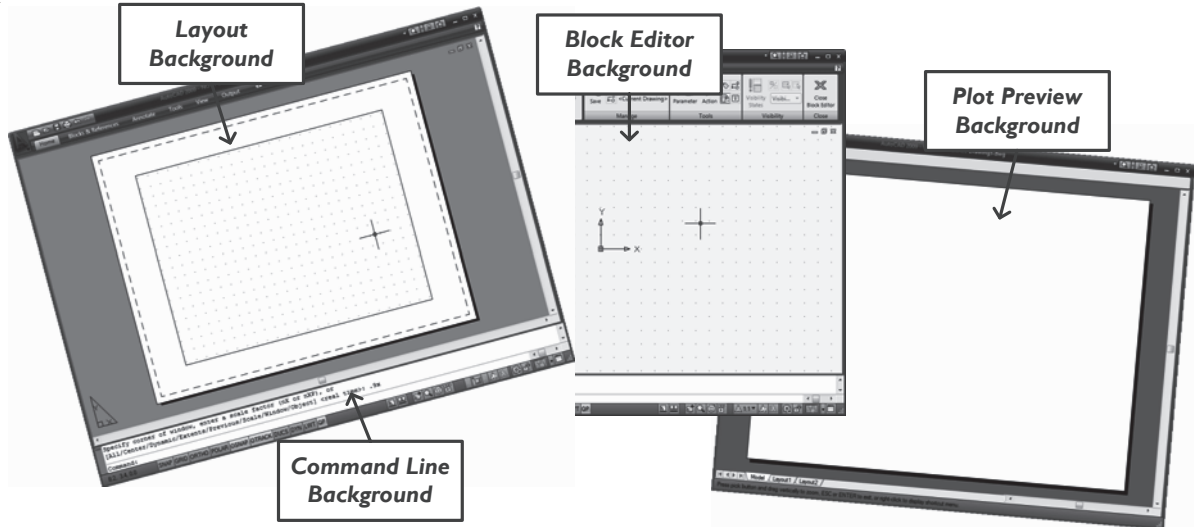
In the “3D Modeling” workspace, AutoCAD displays the drawing area of model space in shades of gray to provide a sense of depth. Notice that the cursor and UCS icon are 3D and colored. (If your copy of AutoCAD does not look like the figure below, start a new drawing with the *acad3d.dwt* template file.)



Other differences in the 3D workspace include grid lines instead of dots, perspective mode instead of parallel projection, the Dashboard, fewer toolbars, and no scroll bars.

Changing Interface Colors

AutoCAD allows you to change the background color of the drawing area and many other parts of its user interface, as illustrated below.

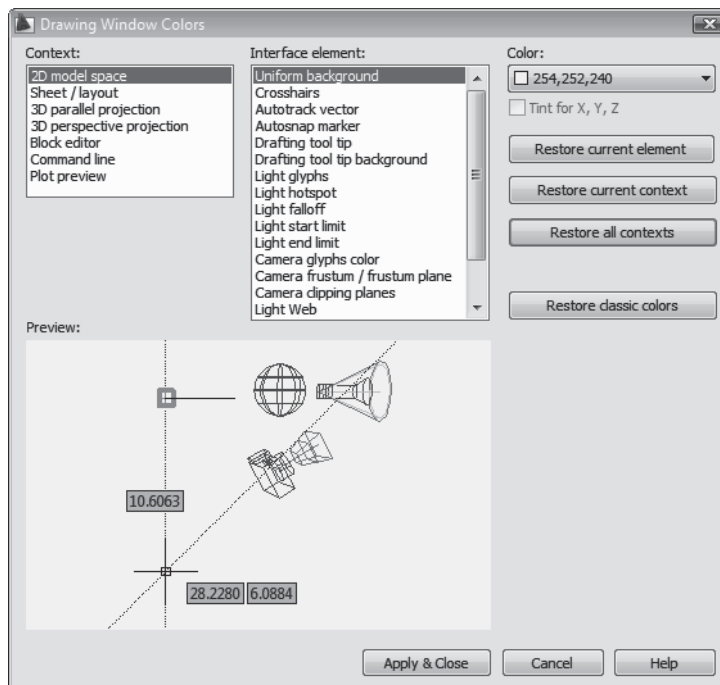


TUTORIAL: CHANGING COLORS

1. To change the colors of the AutoCAD user interface, start the **OPTIONS** command:
 - In the application menu, click the **Options** button.
 - At the 'Command:' prompt, enter the **options** command.

Command: **options** (Press ENTER.)

2. In the Options dialog box, choose the **Display** tab, and then click **Colors**.
AutoCAD displays the Drawing Window Colors dialog box.





Note This dialog box has two important buttons that reset all colors:

Restore All Contexts resets all colors to their defaults.

Restore Classic Colors matches the color scheme of AutoCAD 2008 and earlier. It turns the background color of model space to black, and resets all other colors to their defaults.

3. From the **Context** column, select a user interface “context,” such as **2D model space**.
4. From the **Interface Element** column, select an element, such as **Uniform background**.
5. From the **Color** droplist, assign a color to the element.
(If the basic eight colors are insufficient, click **Select Color**. AutoCAD displays the Select Color dialog box, which accesses the full palette of 16.7 million colors.)
6. If you mess up, click **Restore current element** to change the element back to its default color. Or, choose **Restore current context** to change all elements back to their default colors.
7. Click **Apply & Close** to see the change in colors.
8. Click **OK** to exit the Options dialog box.

0.0 UNITS

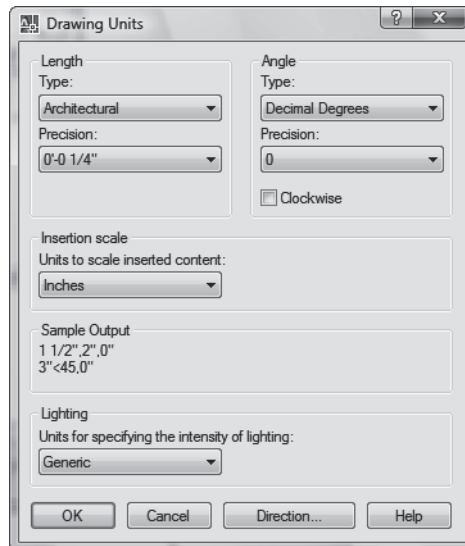
The UNITS command specifies the units of measurement, angles, and precision.

If you choose not to use the Advanced Wizard, set drawing units with this command.

TUTORIAL: SETTING UNITS

1. To set the units for drawings, start the **UNITS** command:
 - At the ‘Command:’ prompt, enter the **units** command.
Command: **units** (Press ENTER.)
 - Alternatively, enter the aliases **un** or **ddunits** (the old name for this command) at the ‘Command:’ prompt.

In all cases, AutoCAD displays the Units dialog box.



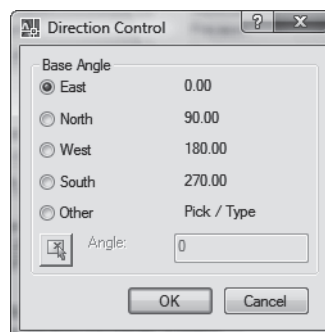
2. In the **Length** area, select the type of unit and precision. To learn more about the units used by AutoCAD, see the discussion earlier in this chapter for the Advanced Wizard.

Watch the Sample Output area for examples of the units you select.

3. In the **Angle** area, select the type and precision of angles.
4. In the **Insertion scale** area (named “Drag-and-drop scale” in earlier releases of AutoCAD), select the units for blocks dragged into drawings from the DesignCenter, Tool palettes, or from a Web site using i-drop:

Unit	Equivalent
Microinches	0.000001 inches
Mils	0.001 inches
Yards	3 feet
Inches	25.4mm
Feet	12 inches
Miles	5,280 feet
Angstroms	0.1 nanometers
Nanometers	10E-9 meters
Microns	10E-6 meters
Millimeters	0.0393 inches
Centimeters	10 mm
Decimeters	0.1 meter
Meters	100 cm
Kilometers	1000 m
Decameters	10 meters
Hectometers	100 meters
Gigameters	10E+9 meters
Astronomical Units	149.597E+8 kilometers
Light Years	9.4605E+9 kilometers
Parsecs	3.26 light years
Unitless	Inserted blocks are not scaled; units match the drawing's units.

5. To change the direction of angle measurement, click **Direction**. AutoCAD displays the Direction Control dialog box.



6. Select a direction for the angle. If you wish to specify an angle that isn't listed, select **Other**, and then enter the angle in the **Angle** text box.

Alternatively, choose the **Pick an Angle** button to pick the angle in the drawing:

Pick angle: *(Pick a point in the drawing.)*

Specify second point: *(Pick a second point.)*

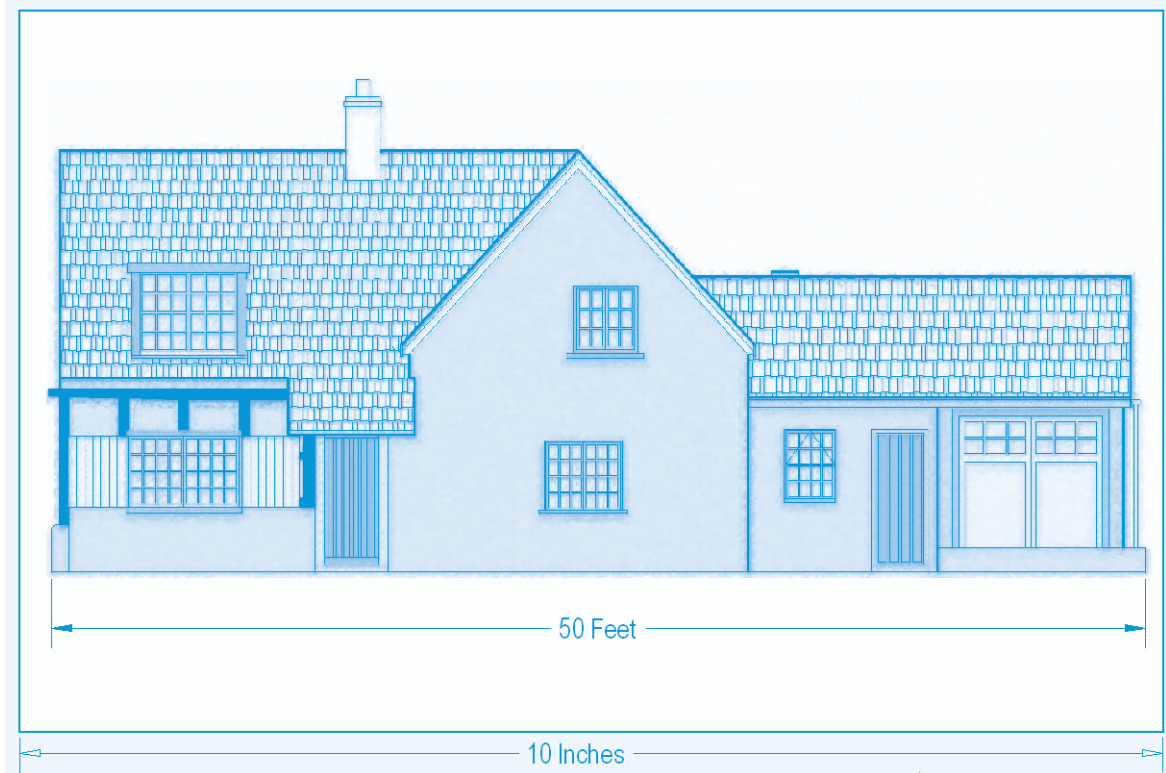
7. Click **OK** to dismiss the Direction Control dialog box.

Click **OK** to dismiss the Drawing Units dialog box.

8. Move the cursor, and watch the coordinate display on the status bar. It should match the units you selected.

SETTING SCALE FACTORS

One of the hardest concepts for the new CAD user to grasp is that of *scale*. Think of when you sketch a picture of your house on a piece of paper. You draw the house small enough to fit the paper. (There isn't a piece of paper big enough for you to draw the house full size! "I don't know about that," rejoins the technical editor. "I've seen paper rolls 50' wide and several thousand feet long. A full-size house drawing would be difficult to fold up, however, and carry under your arm.")



A typical house is 50 feet long. To fit a drawing of it on a 10"-wide sheet of paper, you sketch the house about 60 times smaller. Where does "60" come from? The math works out like this:

Convert house width from feet to inches: 50 feet x 12 inches/foot = 600 inches.

Divide the house size (600 inches) by the paper size (10 inches) = 60.

The scale factor is 1:60. One inch on the drawing is 60 inches on the house.

Here are additional examples of scale factors:

A truck is 20 feet long. The paper is 10 inches wide. The scale factor works out to 1:24.

The sailboat is 30 m long. The paper is 1 m wide. The scale factor is 1:30. (It's much easier in metric!)

The gear is 1 inch in diameter. The paper is 8 inches wide. The scale factor is 8:1. (Scale factor numbers are "reversed" when objects are smaller than the paper.)

In CAD, scaling is done late in the process. Instead of drawing the house to scale, you create the CAD drawing full-size. That means the 50-foot long house is drawn 50 feet long; the one-inch gear is drawn one inch in diameter. That's much easier, isn't it? (AutoCAD can handle drawings of extreme size. The technical editor notes that the largest object would be a circle with a radius of 1 000 000 000 000 000 000 000 E+99 units.)

You could draw the entire known universe — full size — if you want. Full size is shown as 1:1. There is no need to work out scale factors — or is there?

Scale comes into the scene when you plot your drawing. That big drawing, whether of your house or the solar system, must be made small enough to fit the paper it is plotted on. Thus, scale factor comes into play during the PLOT command, where you specify that AutoCAD should print the house drawing 60 times smaller to fit the paper.

Scale-dependent Objects

There is, however, a catch. Some items in the drawing cannot be 60 times smaller. These items include text, hatch patterns, linetypes, and dimensions. These are called *scale-dependent*. If AutoCAD plots the text 60 times smaller, you would not be able to read it. The solution is to do the inverse: draw the text 60 times larger. When it is plotted, the text becomes the correct size.

You may be puzzled. How big should text be when plotted on a drawing? The standard in drafting is to draw text 1/8" tall for “normal” sizes. When you place text in your house drawing, specify a height of 7.5" tall (=1/8" x 60). This may seem much too high to you, but trust me: when plotted, it looks correct.

You must apply the scale factor to other scale-dependent objects as well, including dimensions, linetypes, and hatch patterns. Scale factors for hatch patterns, linetypes, and dimensions are stored in these system variables:

System Variable	Scale Factor
HPSCALE	Hatch patterns.
LTSCALE	Linetype patterns.
DIMSCALE	Dimension text height, arrowhead size, and leaders; does not affect dimension values.

To change the scale factor, enter the name of the related system variable. For example, to set the hatch pattern scale to 60:

Command: **hpscale**

Enter new value for HPSCALE <1.0000>: **60**

Fortunately, you can use the same scale factor for all. For text, you specify a scaled height during the TEXT command:

Command: **text**

Current text style: "TECHNICLIGHT" Text height: 0'-2"

Specify start point of text or [Justify/Style]: (Pick a point.)

Specify height <0'-2">: **7.5**

Alternatively, use the STYLE command to create a text style with a fixed height of 7.5". If the text is the wrong size, you can use the SCALETEXT command to change it.



Note Be aware of this inconsistency: **HPSCALE** and **DIMSCALE** do not apply to objects already in the drawing, but **LTSCALE** does.

Annotation Scaling

Until recently, there was no master scale factor that applied to all scale-dependent objects, and there was no global scale factor for text at all. AutoCAD 2008 introduced *annotation scaling*, a way to ensure that text, dimensions, linetypes, and hatch patterns are displayed and plotted at the correct size.

Annotation scaling works like this: when the annotative scale of objects matches the view and plot scales, then the objects appear; when not, they do not appear. If the drawing is plotted at several sizes, then you need to assign several annotation scale factors to each object. (Fortunately, AutoCAD makes this automatic.) When multiple annotation scales are assigned to objects, AutoCAD takes care of automatically selecting the representation that appears at the correct scale.

Annotation is a property of the following objects: text, mtext, fields, dimensions, tolerances, attributes, leaders, qleaders, multileaders, and hatch patterns. In addition, blocks and dynamic blocks can have annotation scaling.

Linetypes are handled differently. They don't have individual annotative scales; instead, the new **MSLTTYPE** system variable forces all linetypes in the drawing to take on the current annotative scale factor.

You learn more about annotation scaling in Chapter 10, "Placing and Editing Text," as well as in other chapters that deal with scale-dependent objects (hatches, dimensions, and so on).

SAVE AND SAVEAS

The **SAVE** and **SAVEAS** commands operate identically — both display the Save Drawing As dialog box.

The purpose of the commands is to save drawings by other names, and in a few other formats. You may want to give drawings different names when you open them for editing, but don't want to save the changes to the original file. These commands also allow you to save the drawings in other folders, on other drives, and over networks on other computers.

Drawing files created by AutoCAD 2010 cannot be read by versions of AutoCAD earlier than 2008, unless the drawings are *translated*. In contrast, AutoCAD 2010 reads just about any drawing created by earlier releases of AutoCAD. For this reason, the **SAVEAS** command includes options to translate drawings to certain earlier formats.

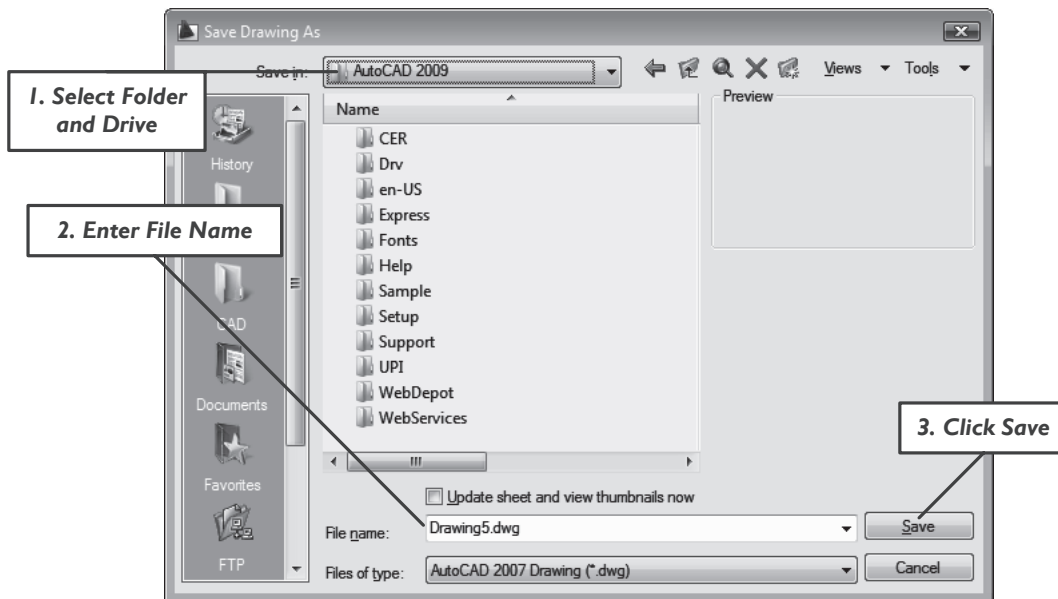
TUTORIAL: SAVING DRAWINGS BY OTHER NAMES

1. To save drawings under different names, start the **SAVEAS** command:
 - From application menu, choose **Save As**.
 - Alternatively, use the **CTRL+SHIFT+S** keyboard shortcut.
 - Or, at the 'Command:' prompt, enter the **saveas** or **save** commands.

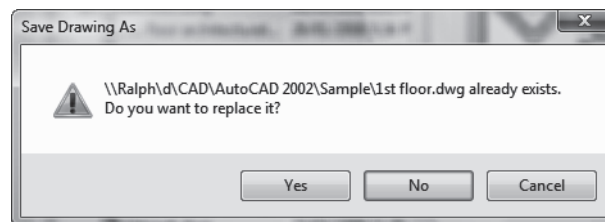
Command: **saveas** (Press ENTER.)

In all cases, AutoCAD displays the Save Drawing As dialog box.

2. In the **Save in** droplist, select the drive and folder in which to save the drawing.
3. In the **File name** text box, enter a different name for the drawing. There is no need to include the **.dwg** extension; AutoCAD adds it automatically.
4. Click **Save**.



When you enter the name of a drawing that already exists in the folder, AutoCAD displays the message box shown below.



5. To replace the existing drawing with the new one, choose **Yes**. If not, choose **No**, and then rename the drawing.

SAVING DRAWINGS: ADDITIONAL METHODS

The SAVEAS command allows you to save drawings on computers connected to networks, including the Internet. It also saves drawings in other formats. The dialog box provides these options:

- **My Network Places** saves drawings over a network.
- **FTP** saves drawings on the Internet.
- **Files of type** saves drawings in other file formats.
- **Options** specifies options for saving .dwg and .dxf files.

Let's look at each option.

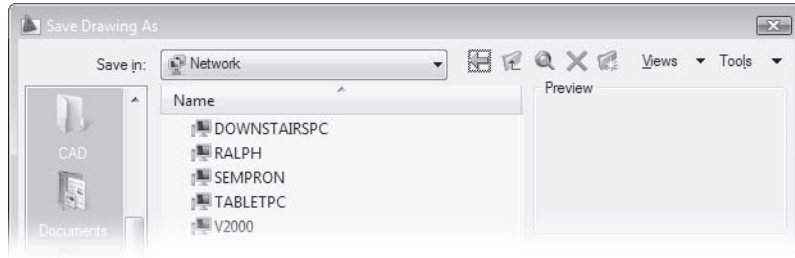
My Network Places

The **My Network Places** option saves drawings to other computers, provided two conditions are met: (1) the computers are connected via a network, and (2) you have access rights.

TUTORIAL: SAVING DRAWINGS ON OTHER COMPUTERS

1. In the Save Drawing As dialog box, click the **Save in** droplist.
2. From the list, select **My Network Places**. Windows displays the drives of computers connected to yours, as well as generic network groupings.
 - *Networked drives* are identified by names similar to "C on Downstairs," where "C" is the name of the hard drive, and "Downstairs" is the name of the computer.

- *Network groupings* are identified by generic names, such as “Entire Network” and “Computers Near Me.” If you regularly access another computer, click **Add Network Place** item to add it to the list.



3. Select a network drive, and then choose the folder in which to save the drawing.
4. Click **Save**.

Depending on the speed of the network, you may notice it takes longer to save drawings through the network.

FTP

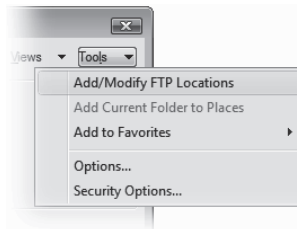
FTP is a method of sending and receiving files over the Internet (short for “file transfer protocol”). You may be familiar with sending files through email as *attachments*. The drawback to using email is that it was not designed to handle very large files. Some email providers limit the attachment size to 5MB or smaller.

FTP allows you to send files of any size, even entire CDs of files — 650MB or more. (FTP is the method used to upload Web pages to Web sites.) The drawback is that FTP first needs to be set up with a user name, password, and other data.

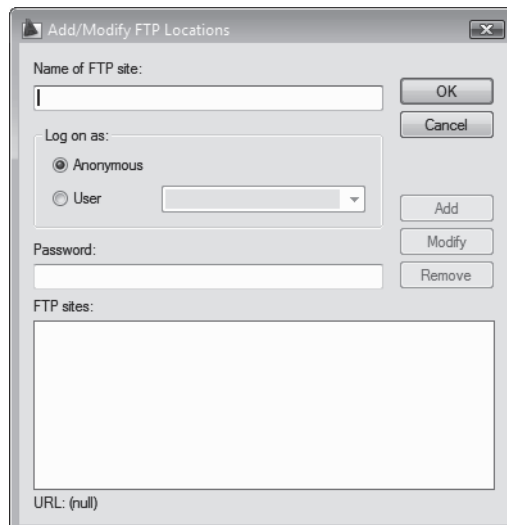
Clicking on **FTP** in the Places list results in an empty list. You first need to add a site.

TUTORIAL: SENDING DRAWINGS BY FTP

1. In the Save Drawing As dialog box, choose **Tools**, and then **Add/Modify FTP Locations**.

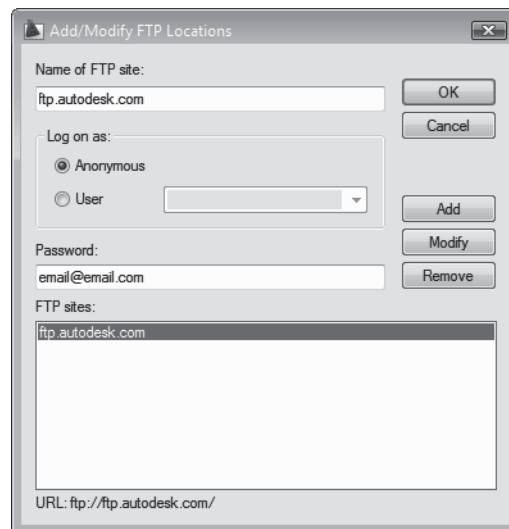


AutoCAD displays the Add/Modify FTP Locations dialog box.

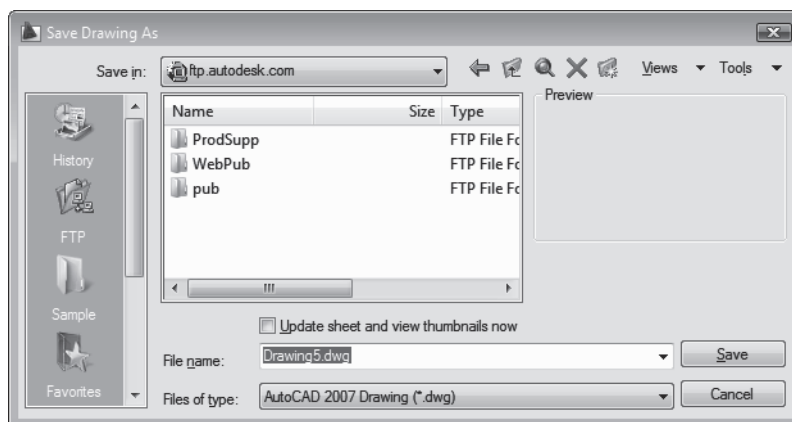


There are two types of FTP sites: anonymous and password.

- *Anonymous* sites allow anyone to access them (also known as “public FTP sites”). You need only your email address as the password; a username is unnecessary.
 - *Password* sites require a username and a password, usually provided by the person running the FTP site.
2. Enter the details for the ftp site. For example, enter the following parameters to access Autodesk’s public FTP site:
 Name of FTP site: **ftp.autodesk.com**
 Logon as: **Anonymous**
 Password: **email@email.com**
 3. Click **Add**.
 4. Click **OK**. Notice that AutoCAD adds the FTP site to the list in the dialog box. The FTP item has been created.

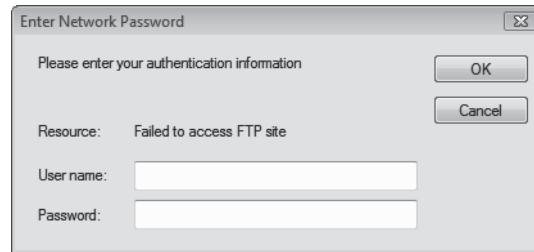


5. To save the drawing, double-click the name of the FTP site (*ftp.autodesk.com*) under the list of FTP Sites.
 Notice that the Save Drawing As dialog box displays a list of folders found on the remote site.



6. Select a folder, and then click **Save**.
 A dialog box shows the progress as AutoCAD transfers the file to the site. Expect it to be slower than saving to disk.

Some sites require a username and password before you can save files. If so, a dialog box appears requesting the information. Fill in the missing information, and click **OK**.



Files of Type

To save the drawing in an earlier release of AutoCAD, or in other AutoCAD formats, choose the **File of type** list box.

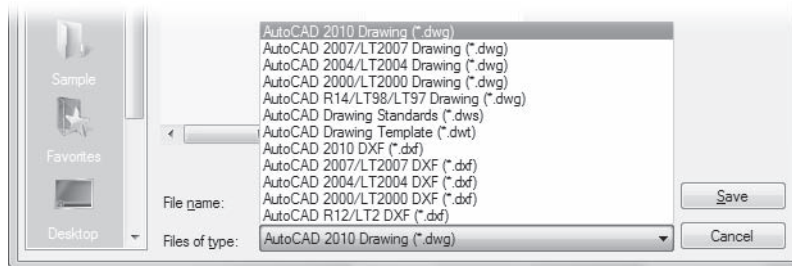
TUTORIAL: SAVING DRAWINGS IN OTHER FORMATS

1. To save drawings in other formats, start the **SAVEAS** command:

Command: **saveas** (Press ENTER.)

AutoCAD displays the Save Drawing As dialog box.

2. In the **Files of type** droplist, select a file format:



- **AutoCAD 2010 (*.dwg)** saves drawings in 2010 format, which can also be read by AutoCAD LT 2010.
- **AutoCAD 2007/LT 2007 (*.dwg)** saves drawings in 2007 format, which can also be read by AutoCAD 2008/9 and AutoCAD LT 2007/8/9.
- **AutoCAD 2004/LT 2004 Drawing (*.dwg)** saves drawings in 2004 format, which can also be read by AutoCAD 2005 and 2006.
- **AutoCAD 2000/LT 2000 Drawing (*.dwg)** saves drawings in 2000 format, which can also be read by AutoCAD 2000i and 2002, as well as by AutoCAD LT 2000i and 2002.
- **AutoCAD R14/LT 98/LT 97 Drawing (*.dwg)** saves drawings in Release 14 format.
- **AutoCAD Drawing Standards (*.dws)** saves drawings as standards files.
- **AutoCAD Drawing Template (*.dwt)** saves drawings as templates.
- **AutoCAD 2010 DXF (*.dxf)** saves drawings in DXF format compatible with AutoCAD 2010. This saves the drawing in a format called DXF (short for “drawing interchange format”).
- **AutoCAD 2007 DXF (*.dxf)** saves drawings in DXF format compatible with AutoCAD 2007 through 2009.
- **AutoCAD 2004 DXF (*.dxf)** saves drawings in DXF format compatible with AutoCAD 2004 through 2006.

- **AutoCAD 2000/LT 2000 DXF (*.dxf)** saves drawings in DXF format compatible with AutoCAD 2000 and 2000i.
 - **AutoCAD R12/LT R2 DXF (*.dxf)** saves the drawing for use with AutoCAD Release 12. To open the drawing in Release 12, Release 11, and AutoCAD LT Release 1 and 2, use the **DXFIN** command.
3. Click **Save**.
- Caution!* Some objects may be lost or changed in translation to older formats.



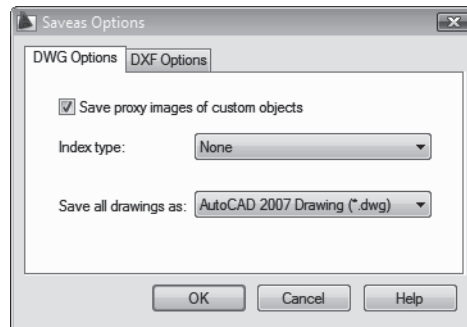
Note To save drawings in...

- TIF, JPG, AND PNG formats, use the **TIFOUT**, **JPGOUT**, and **PNGOUT** commands, respectively.
- PDF format, use the **EXPORTPDF** command.
- ESP (encapsulated PostScript) format, use the **PSOUT** command.
- SLD slide format, use the **MSLIDE** command.
- DWF or DWFX formats, use the **EXPORTDWF** or **EXPORTDWFX** commands, respectively.
- 3D DWF format, use the **3DDWF** command.
- MicroStation V8/V7 format, use the **DGNEXPORT** command.
- WMF, SAT, STL, BMP, or 3DS, use the **EXPORT** command.

Options

The **Options** option allows you to specify how *.dwg* and *.dxf* files are saved.

1. In the Save Drawing As dialog box, choose **Tools**, and then **Options**. AutoCAD displays the Saveas Options dialog box.



2. The **DWG Options** tab has the following settings:
 - ☒ **Save proxy images of custom objects** stores images of *custom objects* in the drawing file. Custom objects are created by ObjectARX programs, and are not understood by AutoCAD when the programs are not present. *Proxy images* are graphical representations of custom objects, which allow you to see, but not edit them.
 - ☐ When off, AutoCAD displays rectangles in place of custom objects. It usually makes sense to leave this option turned on.

Index type specifies whether AutoCAD creates indices when saving drawings. *Indices* improve performance during demand loading, but may increase save time. The options are:

None creates no indices.

Layer loads layers that are on and thawed.

Spatial loads only drawing parts within clipped boundaries.

Layer & Spatial combines both options for optimal partial loading performance.

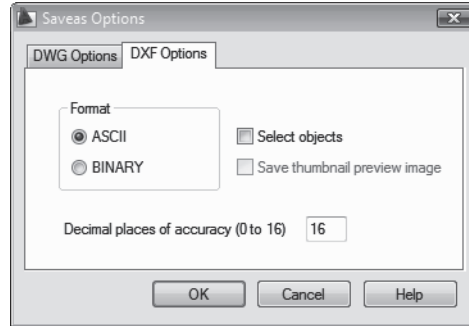
Save all drawings as determines the default format when drawings are saved with the **SAVEAS** and **QSAVE** commands. Keep this set to "AutoCAD 2007 Drawing (*.dwg)" to preserve all features unique to AutoCAD 2007 - 2010. If, however, you regularly exchange

drawings with clients who use earlier releases, you may want to change the setting here.

3. The **DXF Options** tab has these settings:

Format selects either ASCII or binary format:

- ☒ **ASCII** .dxf files are readable by a larger number of non-Autodesk software applications than binary, and also by humans when files are opened in a text editor.
- ☐ **Binary** files are created and read faster than ASCII, and have smaller file sizes.
- ☒ **Select Objects** prompts you to select objects from the drawing.



☐ When option is off, AutoCAD outputs the entire drawing to the .dxf file.

☒ **Save Thumbnail Preview Image** includes a preview image in the .dxf file.

Decimal Places of Accuracy (0 to 16) specifies the accuracy of .dxf files. Higher values, such as 16, create larger files. CNC machines typically work to four decimal places, and get confused when files contain five or more decimal places. This option is available only for ASCII .dxf files; binary files are always saved with 16 decimal places of accuracy.

QSAVE

The QSAVE command (short for “quick save”) saves drawings as .dwg files on disc.

When you start a new drawing, AutoCAD gives it the generic name of *drawing1.dwg*. The first time you use QSAVE, AutoCAD displays the Save Drawing As dialog box, so that you can name the drawing, and file it in the correct folder. Subsequent uses of QSAVE unobtrusively save the drawing without the dialog box.

TUTORIAL: SAVING DRAWINGS

1. To save drawings to disc, start the **QSAVE** command:
 - In the Quick Access toolbar, click the **Save** button.
 - From application menu, choose **Save**.
 - As an alternative, press the **CTRL+S** keyboard shortcut.
 - Or, at the ‘Command:’ prompt, enter the **qsave** command.

Command: **qsave** (Press ENTER.)

2. AutoCAD saves the drawing.

If the drawing’s name is *drawing1.dwg*, then AutoCAD displays the Save Drawing As dialog box.

AUTOMATIC BACKUPS

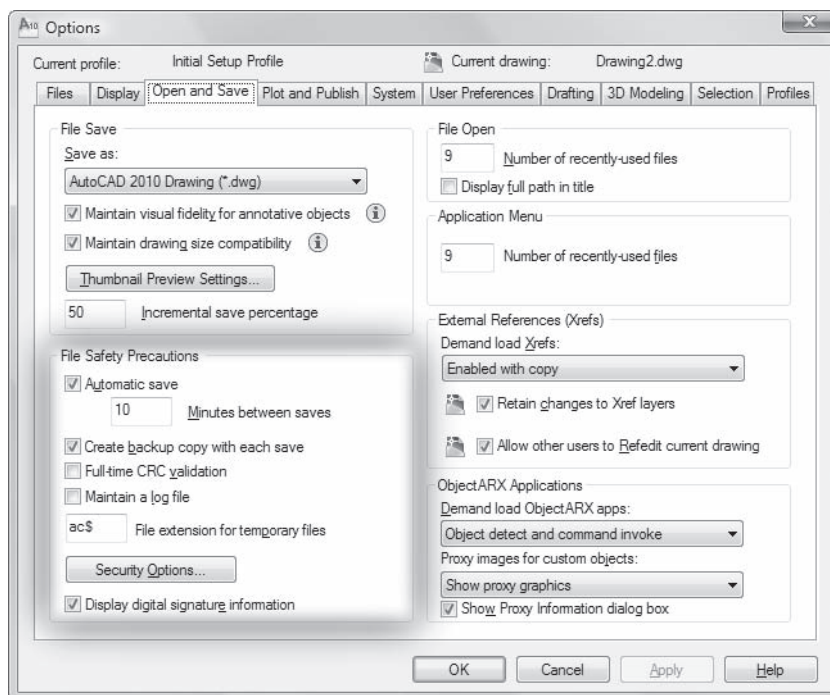
Computers are notorious for *crashing*, where the software stops working for no apparent reason. Even with improvements to Windows XP and Vista, AutoCAD can crash unexpectedly due to *bugs* (accidental errors in the programming code) and conflicts with the operating system. (Technically, computers do not crash; the application software conflicting with the operating system causes the crash.) When this happens, you can lose work.

It is therefore wise to save your work periodically, even if AutoCAD automatically saves to a temporary file. (If you choose to discard your work with the QUIT command, only the part of the drawing changed since the last file save is discarded.)

To guard against loss of work, AutoCAD includes two functions to back up drawings: one makes backup copies of drawing files; the other creates a second set of backup copies at set time intervals. Autodesk warns that it is possible to lose drawing data when the power fails during a save. I recommend plugging your computers into uninterruptable power supplies that provide ten minutes of emergency power, giving you time to save drawings during power outages.

TUTORIAL: MAKING BACKUP COPIES

- To ensure your drawings are backed up, open the Options dialog box:
 - Click the red “A,” and then choose **Options**, from the application menu.
 - At the ‘Command:’ prompt, enter the **options** command.
 Command: **options** (Press ENTER.)
- In either case, AutoCAD displays the Options dialog box. In the Options dialog box, click the **Open and Save** tab.
- Notice the items in the **File Safety Precautions** section:
 - ☒ **Automatic save** turns on the automatic backup facility. *This setting should always be turned on!* The only time to turn it off is when your computer’s hard drive is low on free space — which is rare in this day of huge, cheap hard drives.
 - 10 Minutes between saves** means that AutoCAD automatically backs up the drawing every ten minutes. You can change this value to any number you like. If your drawings are small, and your computer fast, then you may want to set this value to five minutes or less. That way, at most, only five minutes’ work is lost.
 - A setting of 0 (zero) means that no backups are made.



- ☒ **Create backup copy with each save** creates a .bak backup copy each time you use QSAVE and CTRL+S. *This option also should always be turned on, unless the hard drive is low on free space.*
- Click **OK** to save the settings, and exit the dialog box.

AutoCAD starts the **SAVETIME** timer when you first change the drawing, not when you first open the drawing. The timer resets each time you use the **QSAVE**, **SAVE**, and **SAVEAS** commands. This prevents unnecessary hard drive activity.

MAKING BACKUPS: ADDITIONAL METHODS

In addition to instructing AutoCAD to back up copies of your drawings, you can specify the folder in which backup files are stored.

- **SAVEFILEPATH** specifies the folder for storing backup copies.

Let's look at how it works.

SaveFilePath

AutoCAD normally saves backup files in the following folder:

C:\Documents and Settings\username\Local Settings\Temp\

This is the same folder used by all other software on your computer. On my computer, for example, this folder holds 1,873 backup files from a variety of sources — word processors, Web browsers, paint programs, and AutoCAD. With all that clutter, you may prefer to store AutoCAD's backup files in a more suitable location, such as `\autocad 2010\dwgbackups`, so that the backup files are easier to access.

1. Use Windows Explorer to create the `dwgbackups` folder in `\autocad 2010`. (AutoCAD 2010 already has a folder called `\backup` for its own purposes.)
2. In AutoCAD, change the setting of the **SAVEFILEPATH** system variable, as follows:

Command: **savefilepath**

Enter new value for SAVEFILEPATH, or . for none <>: (Enter a new path, such as `"c:\autocad 2010\dwgbackups"`.)

Backup files are now stored in the new folder.

When the path name includes spaces, as does the one shown above, you must surround it with quotation marks.

If the path name is not valid, AutoCAD complains, "Cannot set SAVEFILEPATH to that value. *Invalid*." Ensure the folders exist, and that the full path is correct — *full path* means *all* folder names, from the drive name down to the file folder.

Accessing Backup Files

The first time you save a drawing, AutoCAD adds the `.dwg` extension to the file name. With the second save, AutoCAD renames the file with the `.bak` file extension. Each time you use the **QSAVE** command, AutoCAD updates the backup file.

When AutoCAD crashes, it attempts to rename the `.bak` file as `.bk1`. (If `.bk1` exists, then AutoCAD renames it `.bk2`, and, if necessary, continues with `.bka` and then all the way through to `.bkz`.) This prevents AutoCAD from replacing previous backup files.

Backup files are full drawing files, just with a different file extension. If necessary, you can open the backup files in AutoCAD, following these steps:

1. Use Windows Explorer to copy the `.bak` files to a different folder.
2. Rename the `.bak` files to `.dwg` files.
3. Open the renamed `.dwg` files in AutoCAD.

(These steps are also carried out by the **DRAWINGRECOVERY** command.)

When AutoCAD crashes, it sometimes leaves behind temporary files; if your computer's hard drive is getting low on free space, you can erase these leftover files — provided AutoCAD is not running at the time.

Drawings saved by the automatic backup process are given the extension *.ac\$*.

When drawings are in use, AutoCAD *locks* the drawings to prevent other AutoCAD users from editing them. The lock status is indicated by the presence of *.dwl* and *.dwl2* (drawing lock) files. These files are used by the **WHOHAS** command to determine which user is editing specific drawings.

The *acminidump.dmp* file is created by AutoCAD when it crashes, to help programmers determine the reason.

These files can be erased with Windows Explorer when AutoCAD is not running.

QUIT

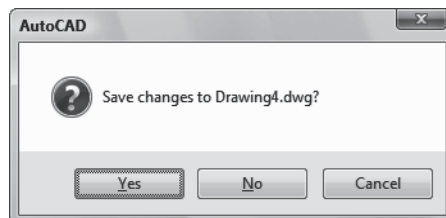
Occasionally you create drawings that you do not wish to keep. To exit the current drawing and discard the changes, use the **QUIT** command.

TUTORIAL: QUITTING AUTOCAD

1. To exit AutoCAD, without saving changes to drawings, start the **QUIT** command:
 - From the application menu, choose **Exit AutoCAD**.
 - At the 'Command:' prompt, enter the **quit** or **exit** commands.
 - Alternatively, use the **CTRL+Q** or **ALT+F4** keyboard shortcuts.

Command: **quit** (Press ENTER.)

2. If nothing has changed in the drawing since it was opened, AutoCAD exits.
In all other cases, AutoCAD displays the dialog box:



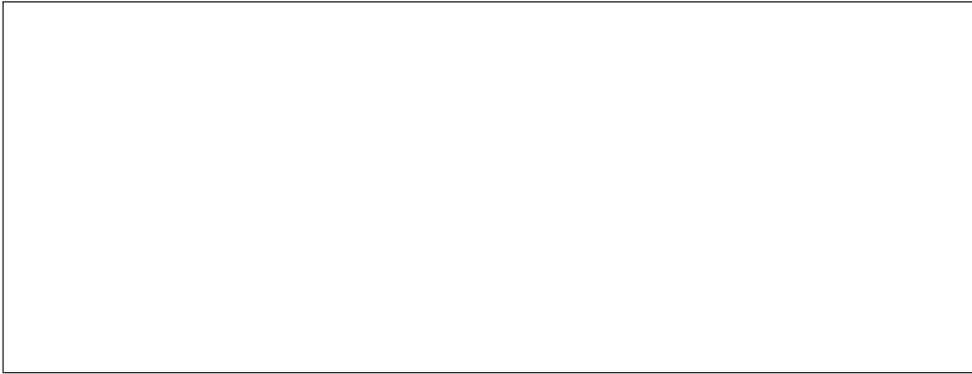
3. To save the drawing, choose **Yes**. AutoCAD displays the Save Drawing As dialog box. Enter a name, and then click **Save**.
To not save the drawing, choose **No**. AutoCAD exits.
To not save the drawing and return to AutoCAD, choose **Cancel**.



Note The **CLOSEALL** command closes all drawings. **QUIT** closes all drawings and exits AutoCAD.

EXERCISES

1. In the following exercises, you start new drawings by a variety of methods.
Start AutoCAD.
Which do you see: a blank drawing, or the Startup dialog box?
If you do not see the Startup dialog box, follow the instruction given in this chapter to turn it on.
2. Once the Startup dialog box is on the screen, click the **Use a Wizard** button.
Select **Advanced Setup**, and then enter the settings for an engineering drawing with four decimal places of precision.
Write down a sample length expressed in the units you set: _____ .
Select Degrees-minutes-seconds with 0 decimal places of precision.
Write down a sample angle expressed in the units you set: _____ .
Select the 0 angle direction as North and clockwise.
Draw a sketch below showing the direction of North and the direction of positive angle measurement:



Specify a drawing size of 11" wide and 17" long.

3. On Quick Access toolbar, click the **New** icon. What happens?
If AutoCAD displays the Create New Drawing dialog box, select the **Use a Template** button.
In either the Create New Drawing dialog box, or the Select Template dialog box, watch the preview window while clicking on the names of template drawings.
When you see a template that looks interesting, open it.
4. In the following exercises, you change the units of measurement.
 - a. From the **Format** menu, select **Units**.
 - b. In the **Length** droplist of the dialog box, select **Fractional Units**.
 - c. Set the precision to **1/16**.
 - d. Click **OK** to exit the dialog box.
 - e. Move the cursor, and watch the coordinates on the status line.
 - f. What do they look like?

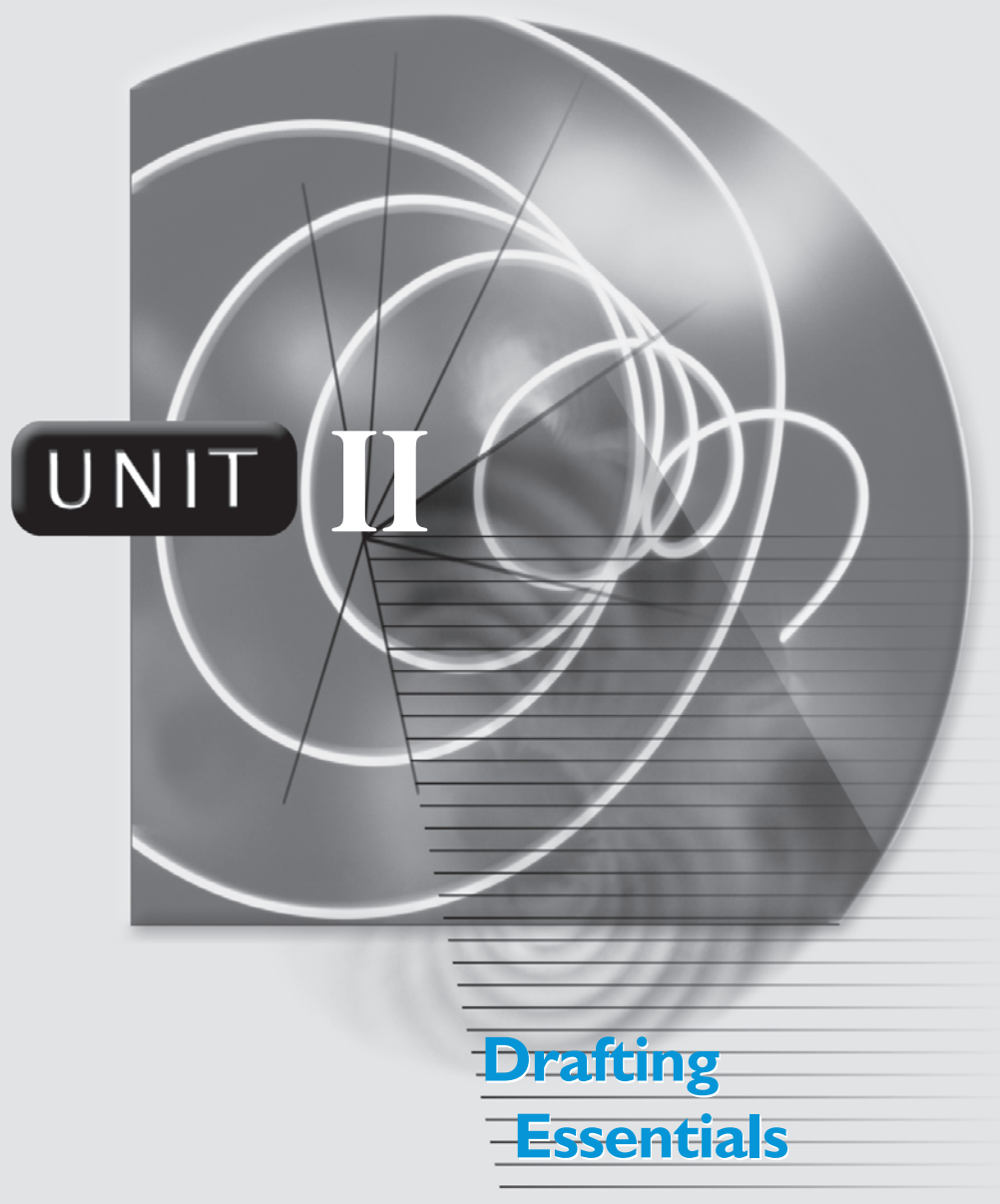
5. Press the spacebar to return to the Units dialog box.
In the **Angle** droplist, select **Surveyor's Units**.
Change the precision to two decimal places.
After exiting the dialog box, start the **LINE** command, and then click a point.
Move the cursor, and watch the status line. What do the coordinates look like?
Move the cursor to the coordinates, and click. Does the display change?
Press **CTRL+D**, and then move the cursor. Is the coordinate display different?
6. Start a new drawing, and then draw a few lines.
Use the **QSAVE** command to save the drawing.
Does AutoCAD ask for a name? If so, enter *save.dwg*.
Repeat the **QSAVE** command. Does AutoCAD ask for a name?
This time, use the **SAVE** command. Does AutoCAD ask for a name? If so, press **Cancel**.
Finally, use the **SAVEAS** command. Does this work identically to the **SAVE** command?
7. If your computer is connected to others through a network, ask your instructor for the following information:
 - Name of another computer.
 - Name of the drive and folder in which to store drawings.Use the **SAVEAS** command to save the drawing from exercise #6 using the network.
8. If your computer has a connection to the Internet, ask your instructor for the following information:
 - Address of the school's FTP site.
 - Username and password (or email address, if an anonymous FTP site).
 - Name of a folder in which to store drawings.Use the **SAVEAS** command to save the drawing from exercise #6 to the Web site using FTP transfer.

CHAPTER REVIEW

1. Is it necessary to add a *.dwg* extension to the drawing name when you save drawing files?
2. What is the danger of saving an AutoCAD 2010 drawing to an earlier version?
3. Which file types does the **OPEN** command open?
4. Write out the meaning of the following file extensions:
 - DWG
 - DXF
 - DWF
 - DWT
5. Does the Select Drawing File dialog box allow you to access drawing files located on the hard drive of your coworkers' computers and at sites located on the Internet?
6. Can the **OPEN** command access Web pages?
7. Is AutoCAD limited to working with just one drawing at a time?
8. Describe how to open more than one drawing in the Select Files dialog box.
9. How can you open a 3D model provided to you in SAT format?
10. Is AutoCAD able to open files other than DWG and DXF?
If so, how?
11. Describe how to rename, move, and delete files with the **OPEN** command.
12. When can you erase a **.ac\$* file?
13. AutoCAD starts up, but does not display the Startup dialog box. Describe how to enable this dialog box.
14. Explain the difference between the **NEW** and **QNEW** commands.
15. What is the importance of the *acad.dwt* file?
16. Describe how *template* drawings are useful.
17. What is the purpose of *wizards* in AutoCAD?
18. Write out the unit of measurement for each example shown below:
 - 1'-2 1/2"
 - 12.3456
 - 2'-2.2"
 - 6.54E+03
 - 43 3/4
19. Which unit of measurement is used for metric drawings?
20. Which unit of measurement does AutoCAD use internally?
21. How many decimal places can AutoCAD display?
When would you not want to work with that many decimal places?
22. When you specify two decimal places, does this restrict the accuracy of AutoCAD's calculations?
23. Write out the angle of measurement for each example shown below:
 - 45.00
 - 45d00'00.00"
 - 50.00g
 - 0.79r
 - N45d0'0.00"E

24. How many *degrees* are there in a circle?
How many *grads*?
How many *radians*?
25. In which direction does AutoCAD think of 0 degrees, by default?
26. In which direction are *positive* angles measured, by default?
27. Can you change the direction of 0 degrees?
28. What does AutoCAD use *limits* for?
29. Does the Select File dialog box allow you to preview drawings?
30. How would you open a drawing, but prevent changes from being saved?
31. What happens when you double-click a drawing file name in the Select Files dialog box?
What happens when you click a drawing file name twice?
32. Explain the meaning of:
Write-protected
Read-only
Read-write
33. Describe how to save drawings that have been opened *read-only*.
34. What are *.bak* files?
Why are they useful?
35. Describe how to open a *.bak* file.
36. How can you guard against losing drawing data from a power outage?
37. What happens when drawings are “saved”?
38. Which command saves drawings quickly?
Can AutoCAD save drawings on its own?
39. Describe the difference between the **QSAVE** and **SAVE** commands.
And the difference between the **SAVE** and **SAVEAS** commands.
40. In the Select File dialog box, displayed by the **OPEN** command, does the **Folders** sidebar provides quick access to:
My Documents
History
FTP
Desktop
Buzzsaw
41. Can you create new folders with the Select Files dialog box?
42. Explain when you would use the **Find** tool?
43. What is the difference between *anonymous* and *private* FTP sites?
44. Describe the difference between the two types of *.dxf* file:
ASCII
Binary
45. Name the command that belongs to each keyboard shortcut:
CTRL+S
CTRL+SHIFT+S
CTRL+O
CTRL+N
CTRL+Q
46. Can you change the background color of the drawing area?
If so, why might you want to?

47. Describe the purpose of the **UNITS** command.
48. How does the **UNIT** command's **Unitless** setting help the operator insert blocks into drawings?
49. Work out the scale factors for drawing these objects on a sheet of paper that is 10 inches wide. Show all your calculations.
 - a. An automobile and trailer 12 feet long.
 - b. A tool shed 10 feet wide.
 - c. A telephone 9 inches long.
50. Work out the scale factors for the following objects drawn on a sheet of paper that is 100 cm wide. Show all your calculations.
 - a. A pen holder 7 cm in diameter.
 - b. A stereo system 42 cm wide.
 - c. A house 16m long.
51. Which system variable sets the scale factor for the following objects:
 - Hatch patterns
 - Linetypes
 - Dimensions
 - Text
52. Explain why text cannot be drawn full-size in drawings scaled at 1:50.
53. How tall should text be drawn in drawings with the following scale factors. (Show all your calculations.)
 - 1:50
 - 1:1
 - 10:1
 - 1:2500
54. If the scale factor for a drawing is 1:50, what should the scale factor be for hatch patterns?
55. If the scale factor for linetypes is 10:10, what should the scale factor be for hatch patterns?
56. Under what conditions can drawings be saved to others' computers?
57. What happens when you use the **SAVE** command to save a drawing, when another drawing of the same name already exists?
58. Describe what happens when you quit AutoCAD without saving the drawing.
59. Explain a benefit and drawback to FTP.
 - What is FTP short for? When might you use FTP?



Drafting Essentials

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Drawing Basic Objects

Now that you know how to set up new drawings, you can begin creating them. Creating drawings with AutoCAD involves using a primary set of commands. These drafting commands construct the objects basic to most drawings — lines, arcs, circles, and so on.

In this chapter, you learn how to drawing basic objects with these commands:

LINE draws line segments.

RECTANG constructs squares and rectangles.

POLYGON constructs regular polygons, from 3 to 1,024 sides.

CIRCLE constructs circles by several methods.

ARC constructs arcs by many methods.

DONUT draws thick and solid-filled circles.

PLINE draws connected lines, arcs, and curves.

ELLIPSE constructs ellipses and elliptical arcs.

POINT draws point objects.

U and **UNDO** reverse the effects of most commands.

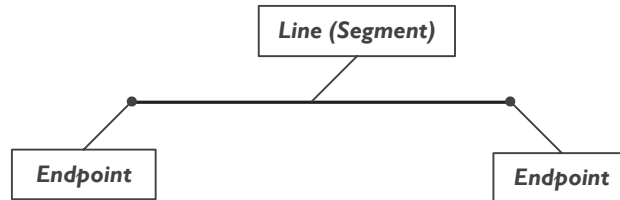
REDO and **MREDO** reverse the effect of the undoing.



The **LINE** command draws line segments.

Constructing lines in drawings is the most basic CAD operation. In AutoCAD, you can draw many types of lines (as listed below) and apply a variety of options to them, but most lines are drawn with the **LINE** command.

A single line is sometimes called a “segment.” At each end, the line has *endpoints*.



Let’s talk about “first points” and “next points,” because you see these terms a lot in AutoCAD. To draw lines and other objects, you first must show AutoCAD the point from which to start. AutoCAD calls this the “first point.” Think of this as the place where you initially put pencil to paper.



Next, determine the point at which the line segment should end. AutoCAD calls this is the “next point.”



The line segment is drawn. AutoCAD then repeats the ‘Specify next point:’ prompt so that you can draw another line segment without the inconvenience of restarting the **LINE** command.



AutoCAD repeats the ‘Specify next point:’ prompt until you press **ESC** to end the command. Each segment is an independent line.



Note AutoCAD has commands that draw different types of lines. You encounter some of these commands in this chapter:

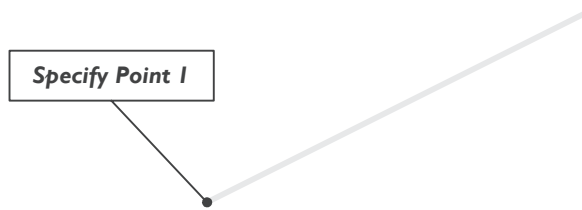
- **MLINE** (multiline) draws as many as 16 parallel lines as a single object.
- **PLINE** (polyline) draws lines in the same manner as the **LINE** command, but a polyline can include arcs, splines, and variable widths. The **3DPOLY** command draws three-dimensional polylines.
- **SKETCH** draws freehand lines.
- **SPLINE** draws splined curves.
- **XLINE** (construction line) and **RAY** draw infinite construction lines (xlines) and semi-infinite construction lines (rays).

TUTORIAL: DRAWING LINE SEGMENTS

- To draw line segments, start the **LINE** command with one of these methods:
 - In the ribbon's 2D Home tab, choose the **Line** button in the Draw panel.
 - Or, at the 'Command:' prompt, enter the **line** command.
Command: **line** (Press ENTER.)
 - Alternatively, enter the **l** alias at the 'Command:' prompt.
- Specify the starting point of the line segment:

Specify first point: (Pick point 1, or specify coordinates.)

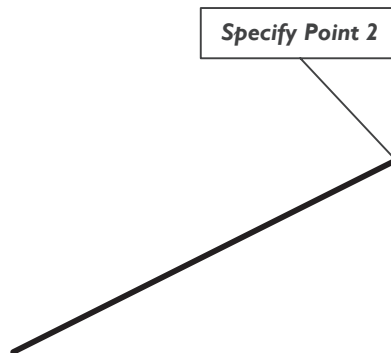
 - With the cursor, pick a point on the screen.
 - Or, at the keyboard, enter x,y coordinates, such as **2,3**.



- Specify the next point(s):

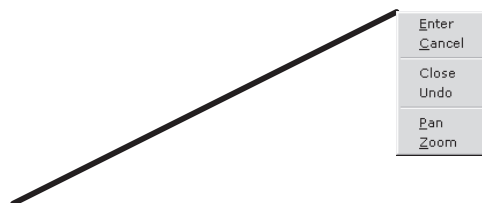
Specify next point or [Undo]: (Pick point 2, or specify coordinates.)

 - Move the cursor, and then pick another point.
 - Or, enter another set of x,y coordinates, such as **3,4**.



- Press **ENTER** to end the command.

Specify next point or [Undo]: (Press ENTER to exit the command.)



Alternatives to pressing **ENTER** include:

- Pressing **ESC**.
 - Right-clicking, and selecting **Enter** from the shortcut menu.
- To repeat the **LINE** command, press **ENTER**.
AutoCAD restarts the **LINE** command and displays "LINE" at the 'Command:' prompt.
Command: **LINE** Specify first point:

6. This time, draw several lines. Notice that each new segment connects precisely at the endpoint of the previous segment. The connection is called a “vertex.”



7. Press **ENTER** to exit the command.

In AutoCAD, pressing **ENTER** ends (most) commands; pressing it again restarts the same command.

DRAWING LINE SEGMENTS: ADDITIONAL METHODS

The **LINE** command provides additional options for drawing lines and ending the command:

- **Close** automatically draws another line to the start of the first one.
- **Undo** undraws the last segment.
- **@** and **<** draw with relative coordinates and angles.
- **ENTER** continues from the last segment.

Some (but not all) of the command’s options are shown on the shortcut menu (below, at left) or in the drafting tooltip (at right).



Note Shortcut menus contain options specific to the command, as well as options that can apply to every command. In the menu illustrated above (at left), four options belong to the **LINE** command: **Enter**, **Cancel**, **Close**, and **Undo**.

The remaining five options apply to any command:

Recent Input — lists the last 20 commands and options you entered.

Snap Overrides — displays a submenu of object snap modes (see a later chapter).

Pan — enters realtime pan, allowing you to shift the drawing around the viewport (see a later chapter).

Zoom — enters realtime zoom, allowing you to change the apparent size of the drawing.

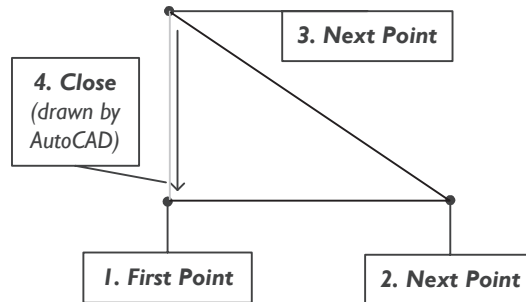
QuickCalc — displays the Quick Calc window, which allows you to carry out calculations, the results of which are used as input for the active command.

Let’s look at each of the options specific to the **LINE** command.

Close

You can use the **LINE** command to construct polygons (multi-sided objects, like the outline of a stop sign). In polygons, the last line segment connects to the first one, but aligning them can be tedious. The **Close** option does this for you automatically. (This option does not appear until you have picked three points, because a minimum of two line segments are needed before a polygon can be created by the third closing segment.)

To see how this works, consider drawing a right angle triangle. The illustration shows the line segment at the final intersection being connected with the **Close** option.



Command: **line**

Specify first point: *(Pick point 1.)*

Specify next point or [Undo]: *(Pick point 2.)*

Specify next point or [Undo]: *(Pick point 3.)*

Specify next point or [Close/Undo]: **c**

Undo

As you draw line segments with the **LINE** command, you may make a mistake. Instead of canceling the command and starting over, use the **Undo** option to “undraw” the last segment. You can undo all the way back to before the first point was placed.

Relative Coordinates and Angles

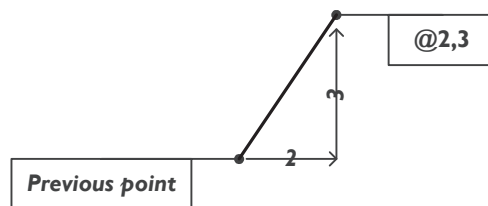
The **@** and **<** symbols allow you to specify relative coordinates and angles. When you prefix x,y-coordinates with **@**, AutoCAD reads the numbers as distances, not coordinates. When you include the **<** symbol, AutoCAD reads the number following it as an angle.

Here are two examples of using relative coordinates:

Command: **line**

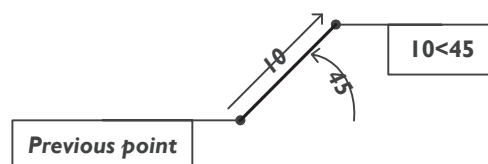
Specify first point: **@2,3**

Draws the line 2 units right and 3 units up from the previous point.



Specify next point or [Undo]: **10<45**

Draws the line 10 units long at a 45-degree angle from the last point.



Continue

When you terminate the **LINE** command, you usually begin another line somewhere else, but sometimes you want to continue from the last segment. To connect precisely, use the **LINE** command's hidden "Continue" option: when you next start the **LINE** command, press **ENTER** at the "Specify first point:" prompt.

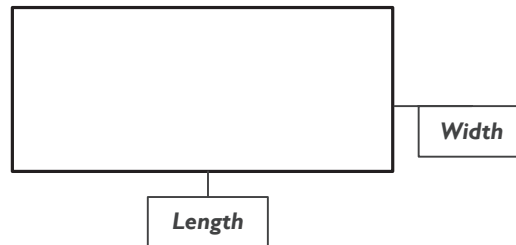
This activates AutoCAD's hidden **Continue** option, and the start point of the new segment is located precisely at the endpoint of the last-drawn segment.



RECTANG

The **RECTANG** command draws rectangles and squares by a variety of methods and in a variety of styles.

This command can draw rectangles with thin or fat lines, tilt them at an angle, and add rounded or cutoff corners. The primary parts of a rectangle are its *length* and *width*:



This command does not draw geometric variations of rectangles, such as parallelograms or rhomboids.

TUTORIAL: DRAWING RECTANGLES

To draw a basic rectangle, pick two points that define the opposite corners of the rectangle.

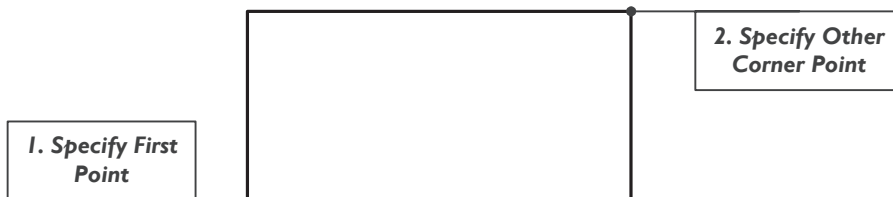
1. Start the **RECTANG** command by one of these methods:

- In the ribbon's 2D Home tab, choose the **Rectangle** button in the Draw panel.
- Or, at the 'Command:' prompt, enter the **rectang** command.

Command: **rectang** (Press **ENTER**.)

- Alternatively, enter the aliases **rec** or **rectangle** at the 'Command:' prompt.

2. Pick a point for one corner of the rectangle, such as the lower-left corner:
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: (Pick point 1.)
3. And pick a point for the opposite corner:
Specify other corner point or [Area/Dimensions/Rotation]: (Pick point 2.)





Notes The sides of the rectangle are parallel to the x and y axes of the current UCS (user-defined coordinate system). To draw rectangles at an angle, use the command's **Rotation** option.

To draw squares, pick two points that create the square. As an alternative, you can use the **POLYGON** command to draw squares.

ADDITIONAL METHODS FOR DRAWING RECTANGLES

The **RECTANG** command provides additional options for drawing rectangles:

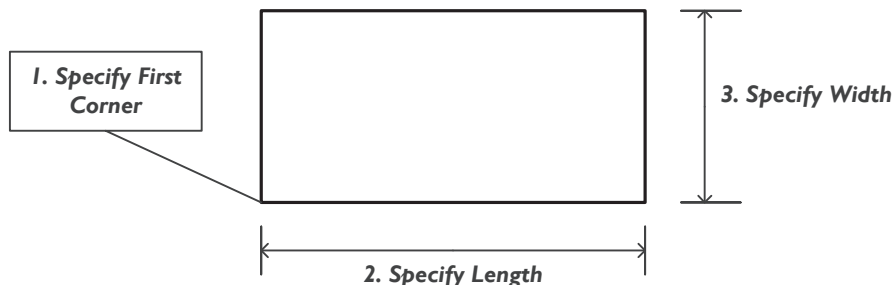
- **Dimensions** specifies the width and height of the rectangle.
- **Area** specifies the area and one side of the rectangle.
- **Rotation** specifies the angle of the rectangle.
- **Width** specifies the width of the four lines making up the rectangle.
- **Elevation** draws the rectangle a specific height above the x,y-plane.
- **Thickness** draws the rectangle with a thickness in the z-direction.
- **Fillet** rounds off the corners of the rectangle.
- **Chamfer** cuts off the corners of the rectangle.

Let's look at each.

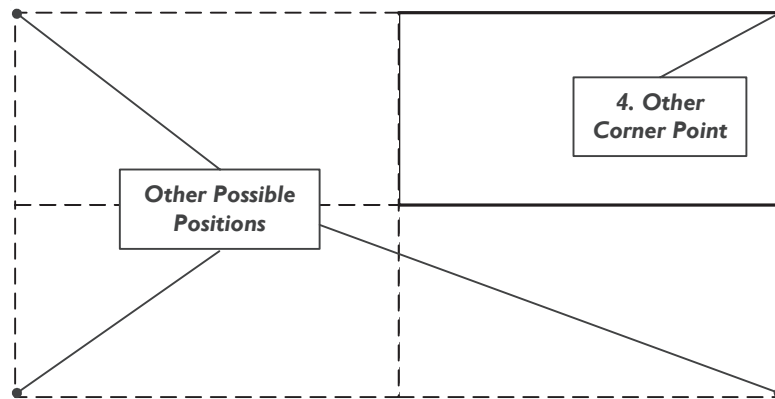
Dimensions

The basic way to draw a rectangle is to specify the points of opposite corners. If you prefer, however, you can specify the rectangle's size by its length and width. In the tutorial below, you draw a 3" x 2" box.

1. Start the **RECTANG** command, pick the starting point, and then enter the **Dimension** option:
 Command: **rectang**
 Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: *(Pick point 1.)*
 Specify other corner point or [Dimensions]: **d**
2. Enter the length, which is in the x-direction (to the right):
 Specify length for rectangles <0.0000>: **3**
3. And enter the width, which is in the y-direction (up):
 Specify width for rectangles <0.0000>: **2**



4. At this point, AutoCAD asks you to specify the orientation of the rectangle. Depending on where you pick the "other corner point," you place the rectangle in one of four positions around the first corner point:
 Specify other corner point or [Dimensions]: *(Move the cursor to position the rectangle, and then pick point 4.)*



Area

The **Area** option specifies the area of the rectangle and the length of one of its sides. After you enter the **Area** option, the command prompts you:

Enter area of rectangle in current units <100>: *(Enter a value.)*

Calculate rectangle dimensions based on [Length/Width] <Length>: *(Type L or W.)*

When you enter **L**, AutoCAD prompts:

Enter rectangle length <10>: *(Enter a value smaller than the area.)*

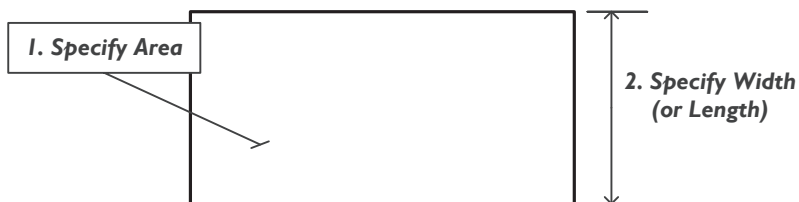
If you enter **W**, AutoCAD prompts:

Enter rectangle width <10>: *(Enter a value smaller than the area.)*

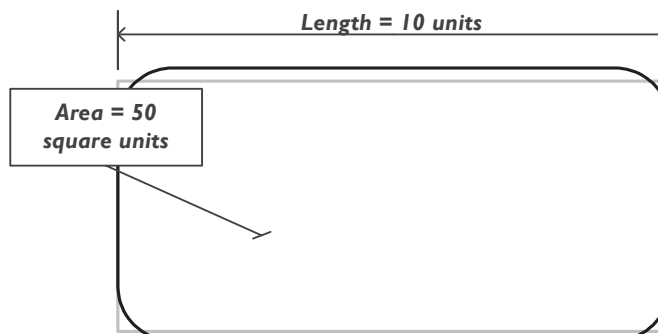
AutoCAD then repeats the earlier prompt:

Specify other corner point or [Area/Dimensions/Rotation]: *(Pick a point or enter an option.)*

You can define the other corner of the rectangle, or specify an option, such as the rotation angle.



When the **Chamfer** or **Fillet** option is active, AutoCAD accounts for the subtracting effect of the chamfers or fillets on the area; that is, the area is made “larger” to compensate. Both rectangles illustrated below have a length of 10 units and an area of 50 square units. The black rectangle seems larger, because the fillets at each corner would otherwise reduce the area.

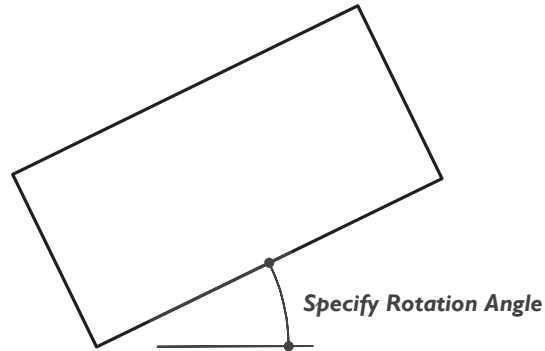


Rotation

The **Rotation** option specifies the angle of the rectangle. AutoCAD prompts you:

Specify rotation angle or [Points] <0>: *(Enter the angle or type P.)*

When you enter the angle, AutoCAD ghosts the rotated rectangle, and then prompts you for the other corner point:

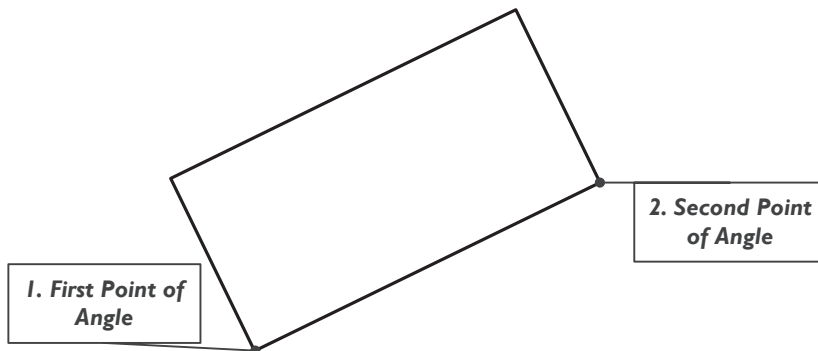


Specify other corner point or [Area/Dimensions/Rotation]: *(Pick a point, or enter an option.)*

When you enter P (for the Points option), AutoCAD prompts you to pick two points that define the angle:

Specify first point: *(Pick a point.)*

Specify second point: *(Pick another point.)*



And then asks for the location of the other corner.

Width

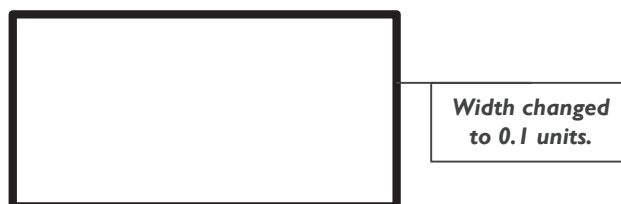
You can specify the width of the four lines making up the rectangle. (Technically, the “lines” are a *polyline*, which are discussed later in this chapter.) Once you set the width, the same value is used by subsequent **RECTANG** commands, until you change it.

To change the width, enter the **Width** option, and then specify a width in units:

Command: **rectang**

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: **w**

Specify line width for rectangles <0.0000>: **.1**



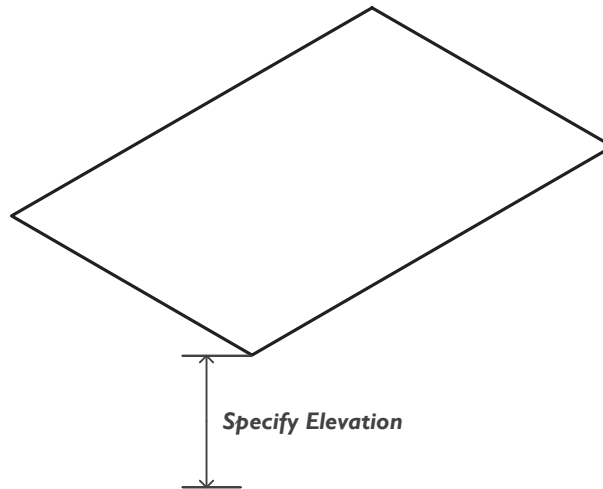
Elevation

You can elevate rectangles by specifying their height above (or below) the x,y-plane — the distance in the z-direction. Once you set the elevation, the same value is used by subsequent **RECTANG** commands, until you change it. To set the elevation, enter the **Elevation** option, and then specify it in units:

Command: **rectang**

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: **e**

Specify the elevation for rectangles <0.0000>: **2**



A positive value for elevation draws the rectangle above the x,y plane. A negative value pushes the rectangle below the x,y-plane.

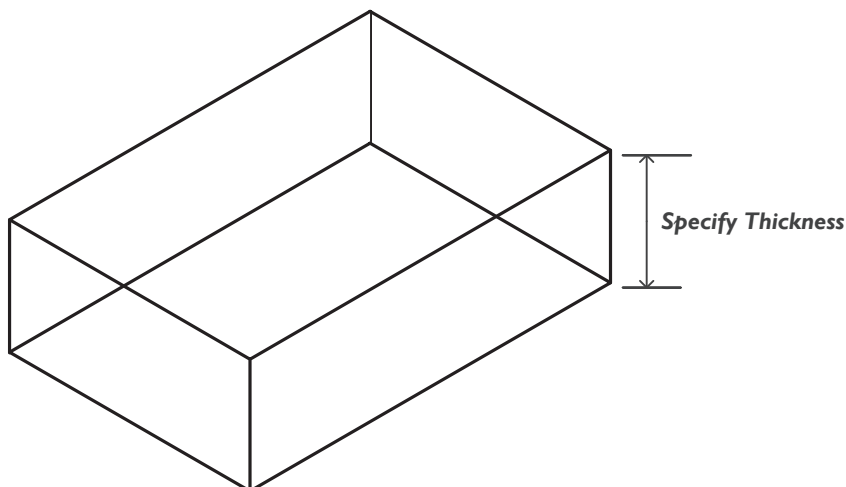
Thickness

You can turn rectangles into 3D box-like objects by specifying a thickness. This is not the recommended way to create 3D boxes, because these lack tops and bottoms. (Use the **BOX** command, to draw solid boxes.) Once the thickness is set, the same value is used by subsequent **RECTANG** commands, until you change it. To set the thickness, enter the **Thickness** option, and then specify it in units:

Command: **rectang**

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: **t**

Specify thickness for rectangles <0.0000>: **2**



A positive value for thickness draws the rectangle upward from the base, while a negative value draws the rectangle downward.

Fillet

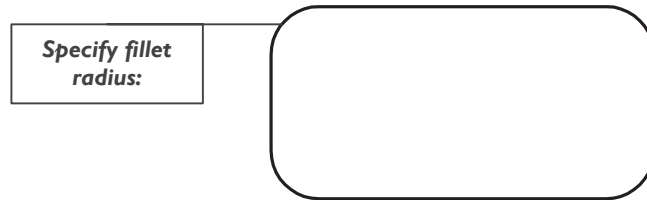
To round the corners of rectangles, enter the **Fillet** option:

Command: **rectang**

Current rectangle modes: Elevation=2.0000 Thickness=2.0000 Width=0.1000

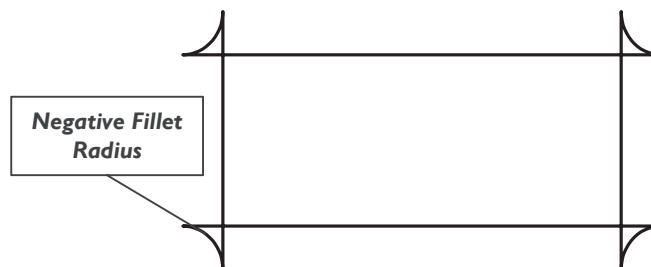
Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: **f**

Specify fillet radius for rectangles <0.0000>: **.25**

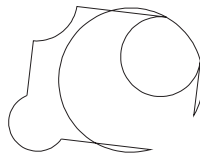


The single fillet radius applies to all four corners; you cannot specify a different radius for each corner. If the fillet radius is too large for the rectangle, AutoCAD does not draw the fillets; instead, the rectangle has square corners. Once you set the fillet, the same value is used until you change it.

You can enter a negative radius, which produces this interesting effect:



After the rectangle is drawn, you can use grips to move the fillets around, such as illustrated below. “But,” warns the technical editor, “just because something *can* be done doesn’t always mean it *should* be done.”



Chamfer

To cut off the corners of the rectangle, enter the **Chamfer** option:

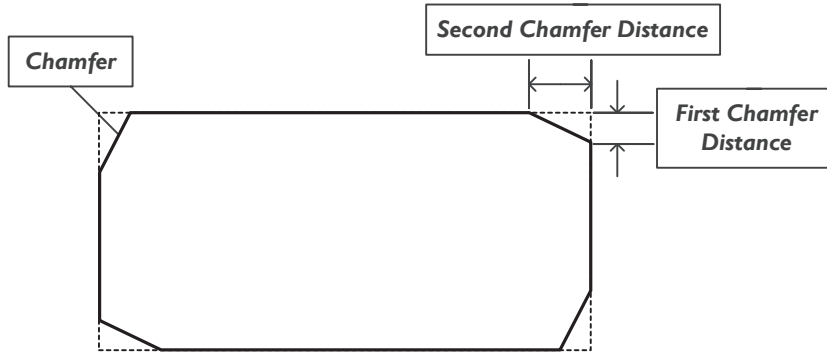
Command: **rectang**

Specify first corner point or [Chamfer/Elevation/Fillet/Thickness/Width]: **c**

Specify first chamfer distance for rectangles <0.2500>: (Press ENTER.)

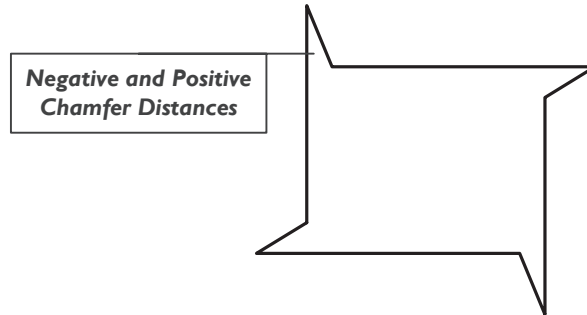
Specify second chamfer distance for rectangles <0.2500>: (Press ENTER.)

The chamfer distances are measured from the corners of the rectangle. You can enter a different value for the first and second chamfer distances, as illustrated below.



The pair of chamfer distances applies to all four corners; you cannot specify a different chamfer for each corner. If the chamfer distances are too large for the rectangle, AutoCAD does not draw the chamfers; instead, the rectangle has square corners. Once you set the chamfers, the same value is used by subsequent **RECTANG** commands, until you change it.

You can enter negative and positive distances, which produces this pinwheel effect:



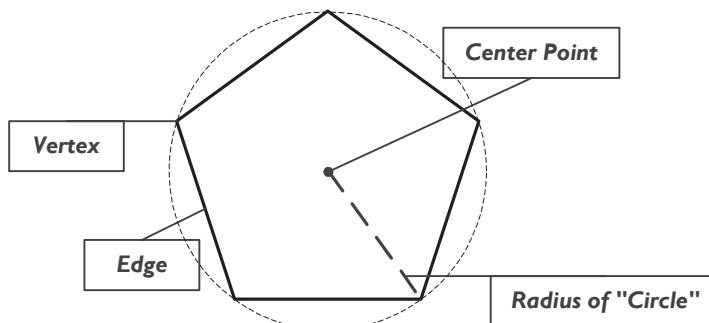
POLYGON

The **POLYGON** command draws regular polygons, from 3 to 1,024 sides.

A *regular polygon* has all the sides the same length; in contrast, an *irregular polygon* has sides of different lengths. With this command, you can draw equilateral triangles, squares, pentagons, hexagons, octagons, and the like. AutoCAD draws polygons from polylines, so you can change their widths with the **PEDIT** command.

Polygons are constructed by any of three methods. By specifying the length of one edge, the polygon is defined, because all lengths are the same. By fitting it inside (“inscribed” within) or outside (“circumscribed” by) an imaginary circle, the polygon’s size is defined by the circle’s radius.

The primary parts of polygons are its vertices, edges, radius of inscribed circle, and center point. as illustrated below:



In drawing regular polygons, the word “circle” is used, because each vertex is the same distance from the center point — just like every point on a circle’s circumference.

This command cannot draw irregular polygons, such as acute triangles or parallelograms; you need to construct these using AutoCAD’s other commands, as described at the end of this chapter.

TUTORIAL: DRAWING POLYGONS

1. To draw regular polygons, start the **POLYGON** command by one of these methods:
 - In the ribbon’s 2D Home tab, choose the **Polygon** button in the Draw panel.
 - Or, at the ‘Command:’ prompt, enter the **polygon** command.
 Command: **polygon** (Press ENTER.)
 - Alternatively, enter the **pol** alias at the ‘Command:’ prompt.
2. Specify the number of sides. For example, enter 3 for a triangle, 4 for a square, 5 for a pentagon, and so on.
 Enter number of sides <4>: *(Enter a value between 3 and 1024.)*
3. Pick a point for the center of the polygon:
 Specify center of polygon or [Edge]: *(Pick a point.)*
4. Decide if the polygon fits inside (is inscribed within) or outside (circumscribes) an imaginary circle:
 Enter an option [Inscribed in circle/Circumscribed about circle] <I>: *(Type I or C.)*
5. Specify the radius of the circle, which determines the size of the polygon:
 Specify radius of circle: *(Enter a radius, or pick a point.)*

ADDITIONAL METHODS FOR DRAWING POLYGONS

The **POLYGON** command provides options for drawing polygons:

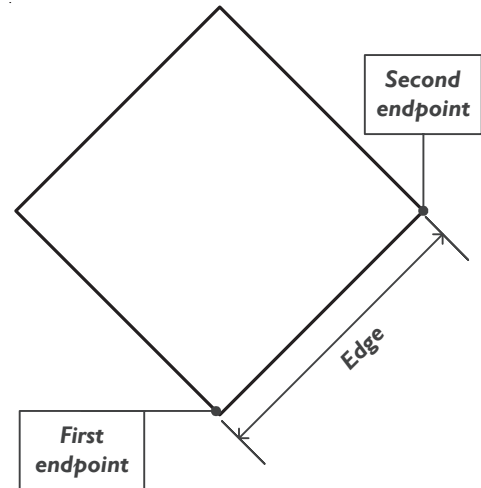
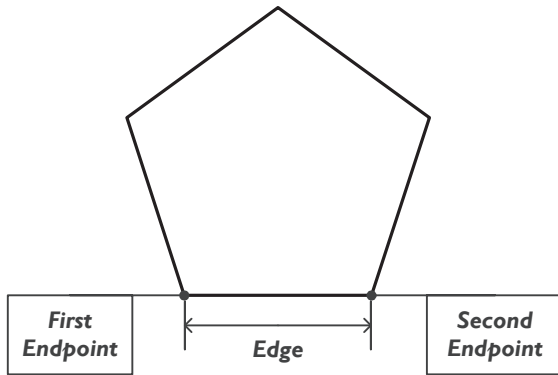
- **Edge** defines the polygon by the length of one of its edges.
- **Inscribed in circle** fits the polygon inside an imaginary circle.
- **Circumscribed about circle** fits the imaginary circle inside the polygon.

Let’s look at each option.

Edge

Perhaps the easiest method is to construct polygons by specifying the length of one edge. Pick two points that define the length of the edge. Here is how to draw a pentagon with the **Edge** option.

1. Start the **POLYGON** command:
 Command: **polygon**
2. To draw a pentagon, enter **5** for the number of sides:
 Enter number of sides <4>: **5**
3. Select the **Edge** option by typing **e**:
 Specify center of polygon or [Edge]: **e**
4. Pick a point (or enter x,y coordinates) for the start of the edge:
 Specify first endpoint of edge: *(Pick a point.)*
5. And pick another point for the end of the edge:
 Specify second endpoint of edge: *(Pick another point.)*



The angle of the two edge points determines the angle of the polygon. For example, to draw a diamond shape, pick the two points at a 45-degree angle (above, right).

Inscribed in Circle

Inscribed polygons are constructed inside a circle of a specified radius. (The circle itself is not drawn, but is imaginary.) The vertices of the polygon fall on the circle. Here is an example of how to draw a triangle inside a circle:

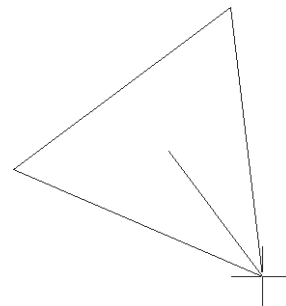
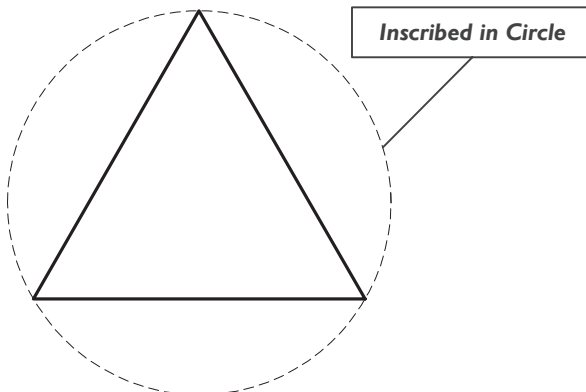
Command: **polygon**

Enter number of sides <4>: **3**

Specify center of polygon or [Edge]: *(Pick a point.)*

Enter an option [Inscribed in circle/Circumscribed about circle] <I>: **i**

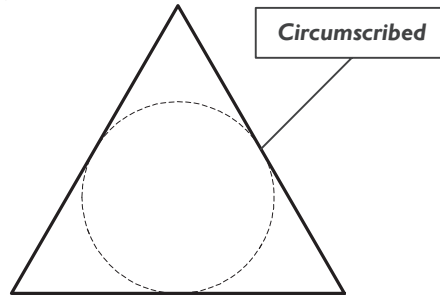
Specify radius of circle: *(Pick a point, or enter a radius.)*



Just as the two points for the **Edge** option determine the rotation of the polygon, so too does the radius specification. As you move the cursor during the “Specify radius of circle” prompt, notice that one vertex moves with the cursor (above, right).

Circumscribed About Circle

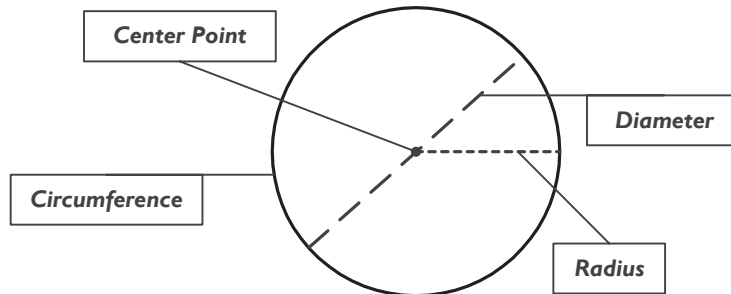
Circumscribed polygons are constructed outside a circle of a specified radius. The midpoints of the polygon’s edges are placed on the circle’s circumference. As with the inscribed option, the rotation of the polygon is determined at the “Specify radius of circle” prompt.



CIRCLE

The **CIRCLE** command draws circles by several methods.

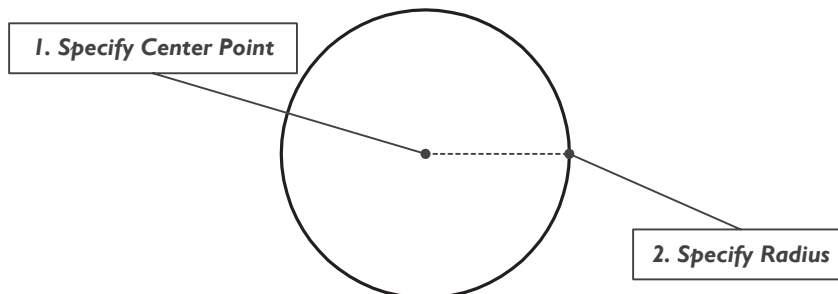
The important parts of the circle are its center point, the radius or diameter, and its circumference.



(This command does not draw isometric circles; instead, use the **Iscircle** option of the **ELLIPSE** command, after turning on **Iso** mode with the **SNAP** command.)

TUTORIAL: DRAWING CIRCLES

1. To draw circles, start the **CIRCLE** command by one of these methods:
 - In the ribbon's 2D Home tab, choose the **Circle** button in the Draw panel.
 - Or, at the 'Command:' prompt, enter the **circle** command:
 Command: **circle** (Press **ENTER**.)
 - Alternatively, enter the **c** alias at the 'Command:' prompt.
2. Pick a point for the center of the circle:
 Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: (Pick point 1, or enter x,y coordinates.)



3. Enter the radius of the circle:
 Specify radius of circle or [Diameter]: (Specify the radius, or pick point 2.)

ADDITIONAL METHODS FOR DRAWING CIRCLES

The **CIRCLE** command provides additional options for drawing circles:

- **Diameter** draws circles based on a center point and diameter.
- **2P** draws circles based on two diameter points.
- **3P** draws circles based on three points along the circumference.
- **Ttr (tan tan radius)** draws circles touching two tangent points and a radius.
- **Tan, Tan, Tan** draws circles touching three tangent points.

Let's look at each option.

Diameter

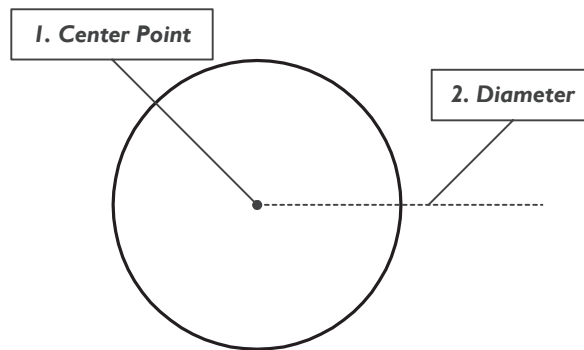
Instead of specifying the circle's center point and radius, you can specify the diameter. (The diameter is twice the radius.)

Command: **circle**

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: *(Pick point 1.)*

Specify radius of circle or [Diameter] <1.0>: **d**

Specify diameter of circle <2.0>: *(Pick point 2, or enter a value for the diameter.)*



2P

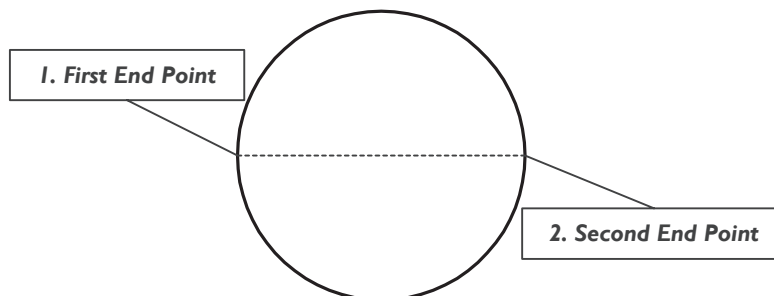
Instead of specifying a center point and diameter, you can specify only the diameter by picking two points on the circumference.

Command: **circle**

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **2p**

Specify first end point of circle's diameter: *(Pick point 1.)*

Specify second end point of circle's diameter: *(Pick point 2.)*



3P

As an alternative, you can pick any three points on the circumference, and AutoCAD constructs the circle.

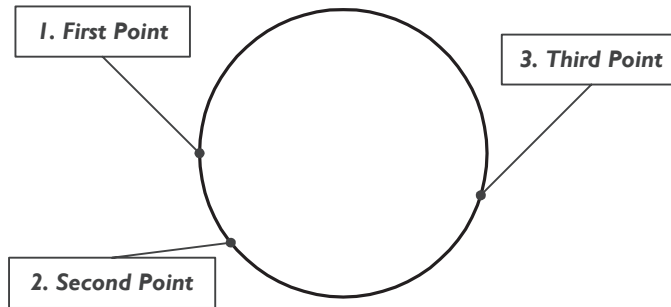
Command: **circle**

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **3p**

Specify first point on circle: *(Pick point 1.)*

Specify second point on circle: *(Pick point 2.)*

Specify third point on circle: *(Pick point 3.)*

**Ttr (tan tan radius)**

Often, you need to draw a circle that's precisely tangent to other objects. The **Ttr** option automatically invokes **Tangent** object snap. (A *tangent* is the point at which the circle touches another object.) You can draw circles tangent to other circles, arcs, lines, polylines, and so on.

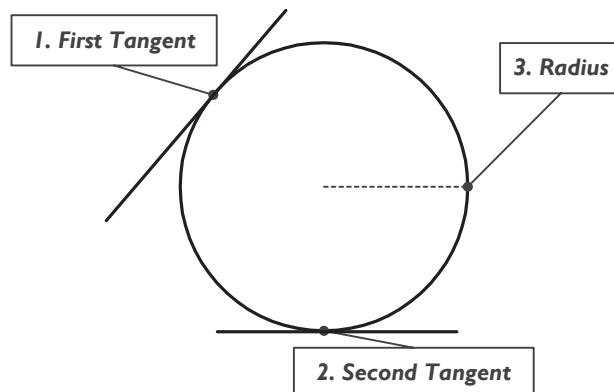
Command: **circle**

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **t**

Specify point on object for first tangent of circle: *(Pick point 1.)*

Specify point on object for second tangent of circle: *(Pick point 2.)*

Specify radius of circle <0.9980>: *(Pick point 3, or enter a value for the radius.)*



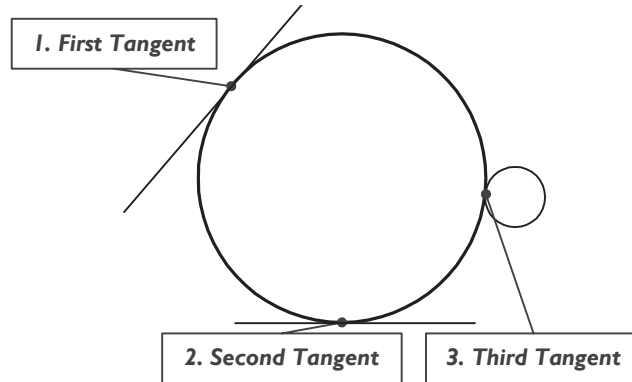
You may find it tricky placing the circle with tangents, because the radius determines where the tangents are placed.

Tan, Tan, Tan

AutoCAD has a sixth way of drawing circles that's not available through the **CIRCLE** command. Instead, you have to access it from the ribbon by selecting **Circle | Tan Tan Tan**. This hidden option constructs circles tangent to three objects. After you select the command from the menu, notice that AutoCAD fills in several options for you, and automatically invokes **Tangent** object snap (displayed as “_tan”).

All you do is pick three objects, as follows:

Command: `_circle` Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: `_3p`
Specify first point on circle: `_tan` to (Pick point 1.)



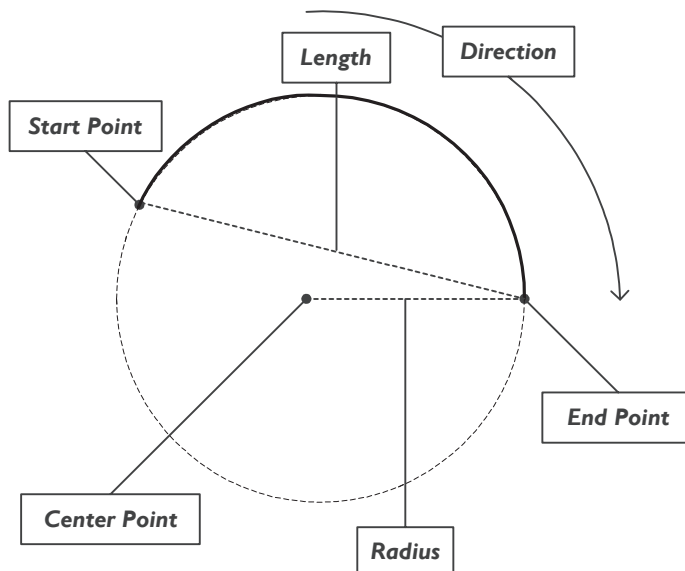
Specify second point on circle: `_tan` to (Pick point 2.)

Specify third point on circle: `_tan` to (Pick point 3.)



The ARC command draws arcs by a variety of methods.

An *arc* is a portion of a circle. Like a circle, an arc has a center point and a radius (or diameter). Like lines, arcs have starting and ending points. The parts of an arc are shown below:



Some disciplines, such as railroad track design, define arcs by *chords*, the straight distance between the start and endpoints. (AutoCAD calls this distance the “length.”)

If AutoCAD does not draw arcs as you expect, it may be because you are trying to draw it clockwise. It can be easy for beginners to forget that AutoCAD usually draws arcs in the *counterclockwise* direction from the starting point. (The exceptions are the three-point arc and the start-end-direction arc.) Sometimes, you may find it easier to draw a circle, and then use the **BREAK** command to create the arc.

AutoCAD constructs arcs by many methods — too many, you might think by the end of this section. This flexibility allows you to place arcs in many different situations.

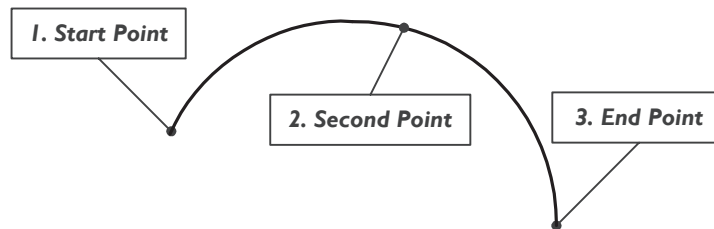
TUTORIAL: DRAWING ARCS

1. To draw arcs, start the **ARC** command by one of these methods:
 - In the ribbon's 2D Home tab, choose the **Arc** button in the Draw panel.
 - Or, at the 'Command:' prompt, enter the **arc** command:

Command: **arc** (Press ENTER.)

- Alternatively, enter the **a** alias at the 'Command:' prompt.
2. Pick the starting point of the arc:
Specify start point of arc or [Center]: (Pick point 1.)
 3. Pick a point that lies on the arc:
Specify second point of arc or [Center/End]: (Pick point 2.)
 4. Pick a point at the end of the arc:
Specify end point of arc: (Pick point 3.)

You can pick points in the drawing, or enter x,y coordinates. The arc drawn in this tutorial is called the "three-point arc." Unlike other arcs, this one is drawn in the direction determined by the first and second points.



ADDITIONAL METHODS FOR DRAWING ARCS

The ARC command provides additional options for drawing arcs. Several of the variations are similar; thus, I have grouped them together:

- **Start, Center, End** constructs arcs from the start, center, and endpoints.
- **Center, Start, End** constructs arcs from the center, start, and endpoints.
- **Start, Center, Angle** constructs arcs from the start and center points, and the included angle.
- **Center, Start, Angle** constructs arcs from the center and start points, and the included angle.
- **Start, End, Angle** constructs arcs from the start and endpoints, and the included angle.
- **Start, Center, Length** constructs arcs from the start and center points, and the chord length.
- **Center, Start, Length** constructs arcs from the start, center, and endpoints.
- **Start, End, Radius** constructs arcs from the start and endpoints, and the radius.
- **Start, End, Direction** constructs arcs from the start and endpoints, and the direction.
- **Continue** continues the arc tangent to a line or another arc.

Let's look at each option.

START, CENTER, END & CENTER, START, END

The most common method of drawing arcs is to specify their start, center, and endpoints. AutoCAD constructs the arc in the counterclockwise direction, from the start and to the endpoint. AutoCAD calculates the arc's radius as the distance from the start point to the center point.

If the arc is too big or looks wrong, it could be you started the arc at the point where it should end. From the **Draw** menu, select **Arcs | Start, Center, End** or **Center, Start, End**.

Command: **arc**

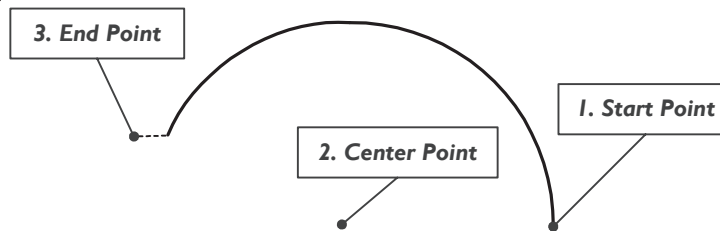
Specify start point of arc or [Center]: *(Pick point 1, or enter x,y coordinates.)*

Specify second point of arc or [Center/End]: **c**

Specify center point of arc: *(Pick point 2, or enter x,y coordinates.)*

Specify end point of arc or [Angle/chord Length]: *(Pick point 3, or enter x,y coordinates.)*

After you pick the center point, notice that AutoCAD ghosts the endpoint as you move the cursor. The endpoint that you pick need not lie on the arc.



START, CENTER, ANGLE & START, END, ANGLE & CENTER, START, ANGLE

Arcs are sometimes specified by their *included angle*. This is the angle formed between the start and endpoints, with the angle's vertex at the arc's center point.

Keep in mind that AutoCAD measures the arc in the counterclockwise direction, starting from the start to the endpoint. When you enter a negative angle, AutoCAD draws the arc clockwise from the start point.

Command: **arc**

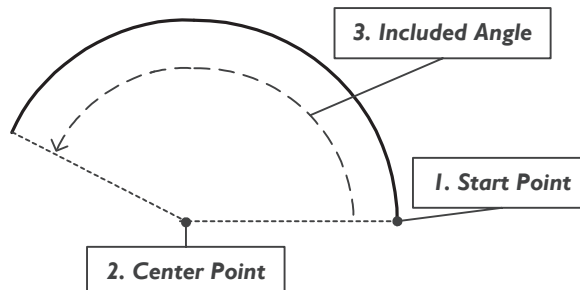
Specify start point of arc or [Center]: *(Pick point 1.)*

Specify second point of arc or [Center/End]: **c**

Specify center point of arc: *(Pick point 2.)*

Specify end point of arc or [Angle/chord Length]: **a**

Specify included angle: *(Enter an angle, 3.)*



START, CENTER, LENGTH & CENTER, START, LENGTH

When you need to draw an arc with a specific chord, use the **Length** option. The length is the distance from the start point to the endpoint.

Command: **arc**

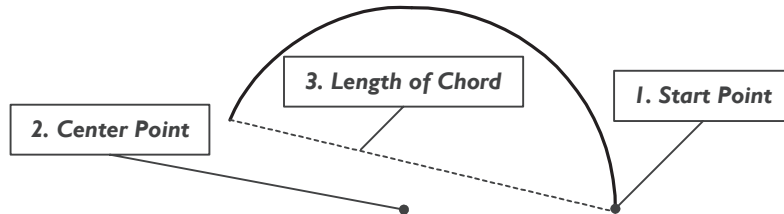
Specify start point of arc or [Center]: *(Pick point 1.)*

Specify second point of arc or [Center/End]: **c**

Specify center point of arc: *(Pick point 2.)*

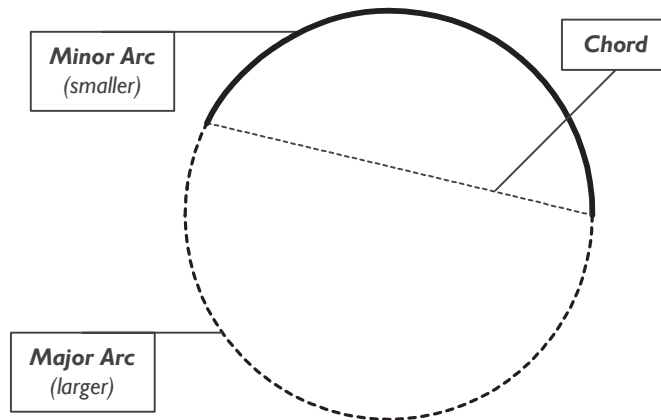
Specify end point of arc or [Angle/chord Length]: **l**

Specify length of chord: *(Enter a length, 3.)*



AutoCAD does not draw the arc if the chord length does not work with the start and center points you pick. When you see “*Invalid*” on the command prompt area, the chord is too long or too short. Note that the center point is *not* usually on the chord.

AutoCAD constructs a *major arc* or a *minor arc*, depending on the chord length. (Given that the chord divides a circle into two parts, the major arc is the larger arc, while the minor arc is the smaller one.) When the chord is positive, AutoCAD draws the minor arc; when negative, the major arc.



START, END, RADIUS

This option is most like drawing a circle: you specify two points and the radius.

Command: **arc**

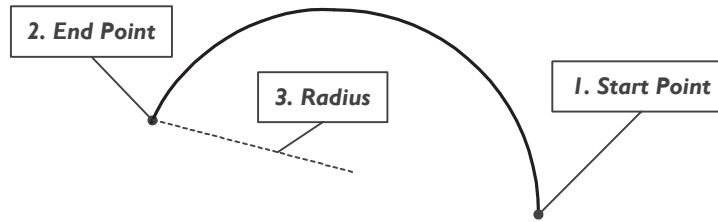
Specify start point of arc or [Center]: *(Pick point 1.)*

Specify second point of arc or [Center/End]: **e**

Specify end point of arc: *(Pick point 2.)*

Specify center point of arc or [Angle/Direction/Radius]: **r**

Specify radius of arc: *(Enter a positive or negative radius, or pick point 3.)*



By default, AutoCAD draws the minor arc. If, however, you enter a negative value for the radius, AutoCAD draws the major arc.

START, END, DIRECTION

This option is perhaps the trickiest to understand. AutoCAD starts the arc tangent in the direction you specify.

Command: **arc**

Specify start point of arc or [Center]: *(Pick point 1.)*

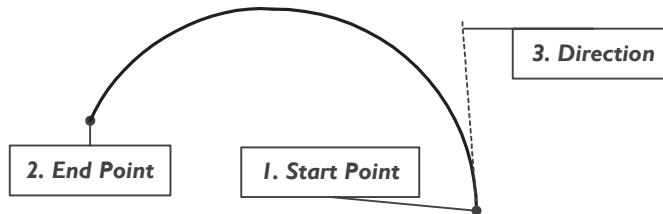
Specify second point of arc or [Center/End]: **e**

Specify end point of arc: *(Pick point 2.)*

Specify center point of arc or [Angle/Direction/Radius]: **d**

Specify tangent direction for the start point of arc: *(Pick point 3.)*

At the “Specify tangent direction” prompt, move the cursor. Notice how AutoCAD ghosts the arc, depending in the cursor’s location.



CONTINUE

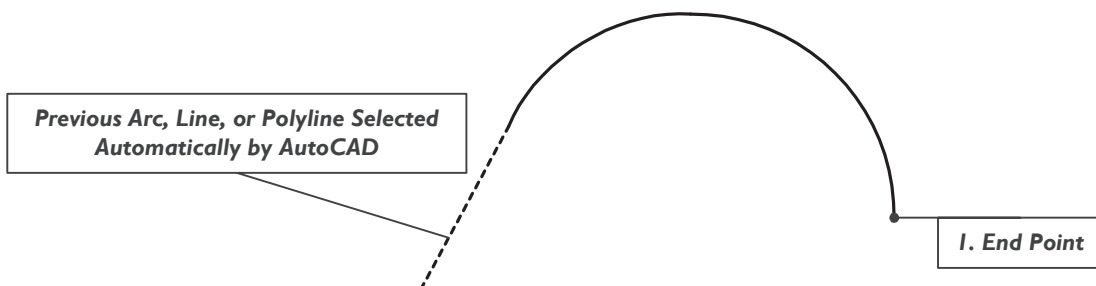
Among the ARC command’s many options, one manages to stay hidden. The **Continue** option is designed to draw an arc tangent to the last-drawn object. Here’s how it works:

Command: **arc**

Specify start point of arc or [Center]: *(Press ENTER.)*

Notice that AutoCAD automatically selects the end of the last line, polyline, or arc as the starting point for the new arc, which is drawn tangent to the last-drawn object. As you move the cursor, AutoCAD ghosts it in the drawing. There is just one option:

Specify end point of arc: *(Pick point 1.)*





Note AutoCAD can convert arcs into circles, and back again. Here's how:

- To convert arcs to circles, use the **JOIN** command's **Close** option.
- To convert circles to arcs, use the **BREAK** command to remove the unwanted part of the circle.



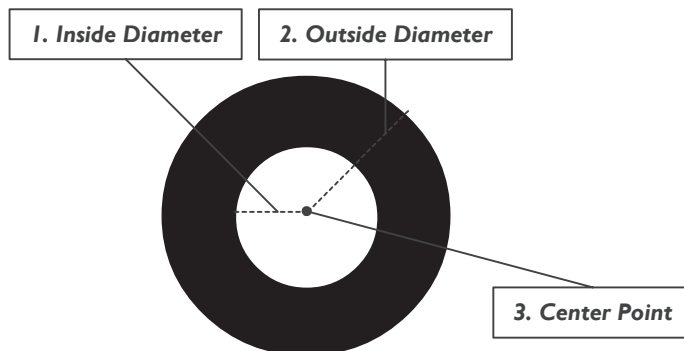
DONUT

The **DONUT** command draws circles with thick walls and solid-filled circles. These kinds of circles are useful for PCB (printed circuit board) designs. Donuts are drawn as polylines.

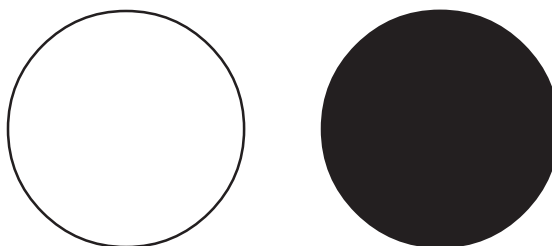
This is one of AutoCAD's few drawing commands that keeps on repeating itself until you press **ESC** or **ENTER**.

TUTORIAL: DRAWING DONUTS

1. To draw donuts, start the **DONUT** command by one of these methods:
 - In the ribbon's 2D Home tab, choose the **Donut** button in the Draw panel.
 - Or, at the 'Command:' prompt, enter the **donut** command:
Command: **donut** (Press **ENTER**.)
 - Alternatively, enter the **doughnut** alias at the 'Command:' prompt.
2. Enter a value for the donut's "hole," its inside diameter:
Specify inside diameter of donut <0.5000>: (Specify value, or pick two points, 1.)
3. Enter a value for the outside of the donut:
Specify outside diameter of donut <1.0000>: (Specify value, or pick two points, 2.)
4. Pick a point to place the donut.
Specify center of donut or <exit>: (Pick point 3.)
5. This command repeats until you exit the command. Press **ENTER** to end it:
Specify center of donut or <exit>: (Press **ENTER** to exit command.)



Note When the inside diameter equals the outside diameter, AutoCAD draws donuts as circles, albeit ones made from polylines. This is a "circle" that can be edited with the **PEdit** command. When the inside diameter is zero, AutoCAD draws solid-filled donuts, as illustrated below.

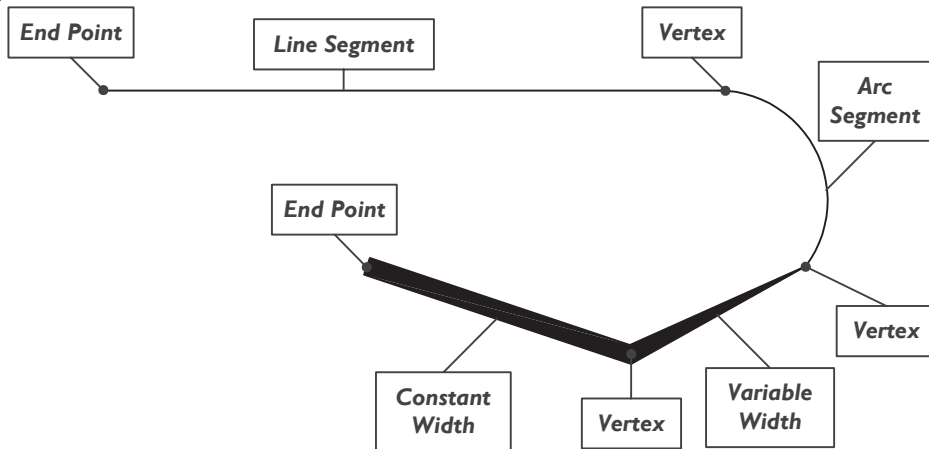


PLINE

The **PLINE** command draws polylines.

We have referred to the “polyline” in some commands earlier, specifically **POLYGON** and **DONUT**. The **PLINE** command draws polylines, perhaps the most unique and flexible object created by any CAD program.

A polyline is a single object that consists of connected lines and arcs. It can be curved, splined, and open or closed (like an irregular polygon). You can specify the width of each segment, or give each segment a tapered width.



(History: Until lineweights were introduced to AutoCAD, polylines were the primary way to draw objects with width.)

TUTORIAL: DRAWING POLYLINES

- To draw polylines, start the **PLINE** command.
 - In the ribbon's 2D Home tab, choose the **Polyline** button in the Draw panel.
 - Or, at the 'Command:' prompt, enter the **pline** command.

Command: **pline** (Press ENTER.)

- Alternatively, enter the **pl** alias at the 'Command:' prompt.
- Pick a point from which to start drawing the polyline:
Specify start point: (Pick a point.)
AutoCAD reminds you of the current line width, which is 0, unless you change it with the **Width** option:
Current line-width is 0.0000
 - Pick the next point, or select an option:
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: (Pick a point, or enter an option.)
 - Continue picking points, and then press **ENTER** to exit the command:
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: (Press ENTER to exit the command.)

ADDITIONAL METHODS FOR DRAWING POLYLINES

The **PLINE** command provides additional options for drawing polylines:

- Undo** removes the previous segment.
- Close** draws a segment to the start point.

- **Arc** switches to arc-drawing mode.
- **Width** specifies the width of the polyline.
- **Halfwidth** specifies the halfwidth.
- **Length** draws a tangent segment of specific length.

Let's look at each option.

Undo

As you draw polyline segments, you may sometimes make a mistake. Instead of canceling the command and starting over, use the **Undo** option to “undraw” the last segment. You can undo all the way back to the first segment.

Close

When you construct irregular polygons using the **PLINE** command, the last segment connects to the first segment. Aligning the ends of segments can be tedious, so the **Close** option automatically performs this for you. The **Close** option appears after you pick three points, because AutoCAD needs a minimum of two segments before it can close the polygon with a third segment.

Arc

The **Arc** option switches the **PLINE** command to arc-drawing mode. This mode has options similar to those of the **ARC** command, with some additions:

Specify next point or [Arc/Halfwidth/Length/Undo/Width]: **a**

[Angle/CEnter/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width]:

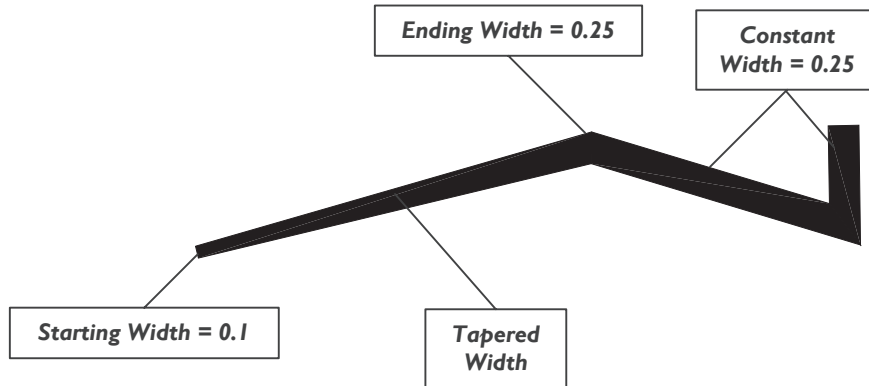
The **Width** and **Halfwidth** options specify the width of the arc segments, as described below. See the **ARC** command for the meaning of the other options.

The **Line** option switches out of arc-drawing mode and back to line drawing mode.

Width

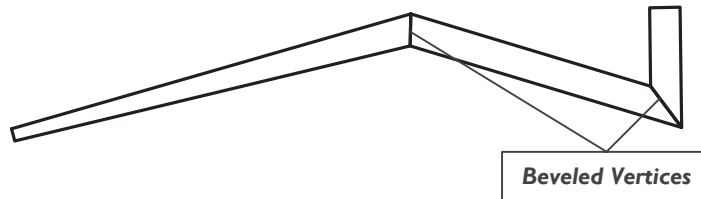
The **Width** option allows you to specify the width of each segment. In addition, you can specify a different starting and ending width, which creates tapers. By default, the width is 0 in new drawings. When changed, AutoCAD remembers the previous width setting.

1. Start the **PLINE** command, and pick a starting point:
Command: **pline**
Specify start point: *(Pick a point.)*
2. Notice the current line width. Enter **w** to access the **Width** option:
Current line-width is 0.1000
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: **w**
3. Enter a value for the “starting width.” This is the width at the starting end of the polyline.
Specify starting width <0.1000>: *(Press ENTER to accept the default, or enter another value.)*
4. Press **ENTER** if you want the ending width to be the same as the starting width. For a taper, enter a different value for the ending width:
Specify ending width <0.1000>: **.25**
5. Pick the next point, or select another option:
Specify next point or [Arc/Halfwidth/Length/Undo/Width]:



Notes When you provide different values for the starting and ending widths, AutoCAD draws a taper but for that segment only! The ending width becomes the next starting width.

When two polyline segments have a width other than 0, AutoCAD automatically bevels the vertex between them. To see the beveled vertices, set the **FILLMODE** system variable to 0, followed by the **REGEN** command.



To select a polyline of width other than 0, you must pick its edge. Picking the centerline of the polyline fails. For some reason, AutoCAD is unable to detect the inside of polylines.

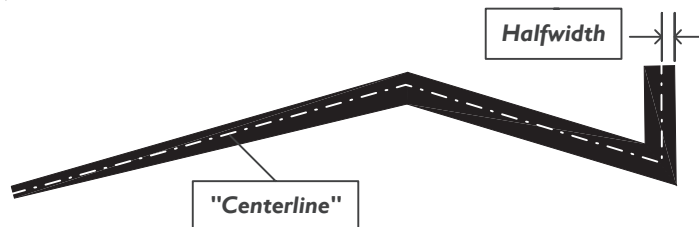
Halfwidth

The **Halfwidth** option is identical to **Width**, except that you specify the width from the edge of the polyline to its centerline.

Specify next point or [Arc/Halfwidth/Length/Undo/Width]: **h**

Specify starting half-width <0.1250>: *(Enter a value, or press ENTER to keep the default.)*

Specify ending half-width <0.1250>: *(Press ENTER for constant width; enter a different value for tapered width.)*

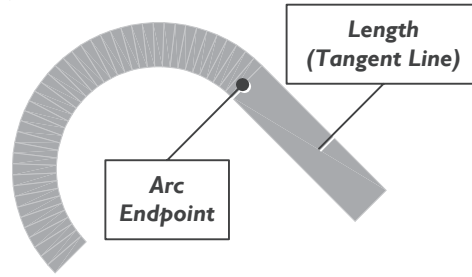


Length

The **Length** option draws the next segment at the same angle as the previous one. You specify the length. This option is most useful when the previous segment is an arc, because the segment is drawn tangentially to the arc's endpoint.

Specify next point or [Arc/Halfwidth/Length/Undo/Width]: **l**

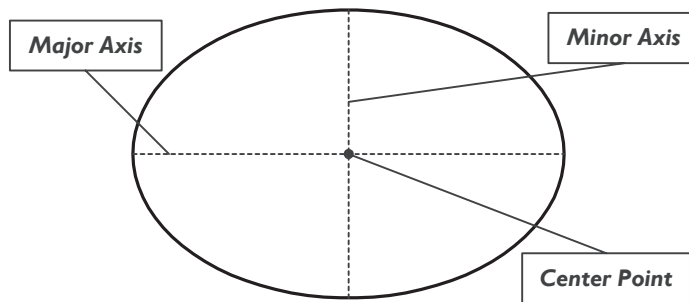
Specify length of line: **3**



ELLIPSE

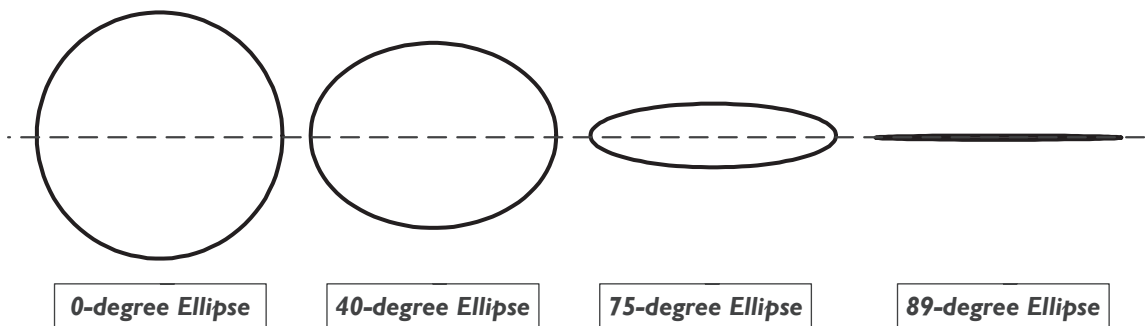
The **ELLIPSE** command draws ellipses and elliptical arcs. When isometric mode is turned on, this command adds the **Isocircle** option for drawing isometric circles.

Ellipses are elongated circles drawn with two diameters called “axes.” The *major axis* is the longer axis; the *minor axis*, the shorter.



An ellipse is sometimes referred to by its *rotation angle*. A circle is an ellipse that has not been rotated: its rotation is zero degrees; the major and minor axes have the same length.

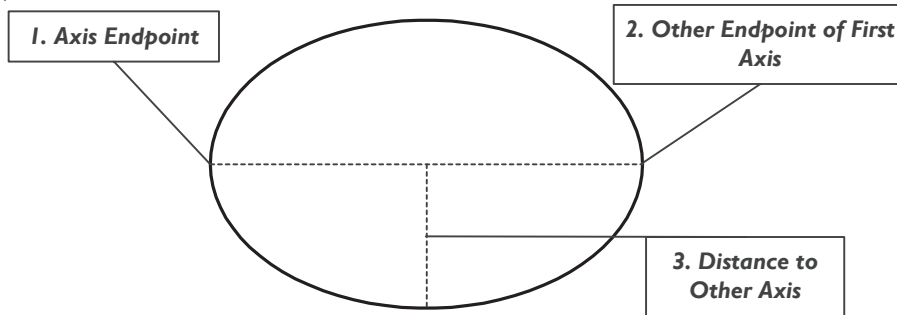
An ellipse is a circle that is rotated, or viewed at an angle. For instance, tilt this page away from you and the circle looks like an ellipse. A 40-degree ellipse is a circle that has been rotated by 40 degrees about the major axis. AutoCAD allows you to specify ellipse rotations between 0.0 (a circle) and 89.4 degrees (a very thin ellipse).



TUTORIAL: DRAWING ELLIPSES

- I. To draw ellipses, start the **ELLIPSE** command.
 - In the ribbon's 2D Home tab, choose the **Ellipse** button in the Draw panel.
 - Or, at the 'Command:' prompt, enter the **ellipse** command.
Command: **ellipse** (Press ENTER.)
 - Alternatively, enter the **el** alias at the 'Command:' prompt.

2. Pick a point to indicate an endpoint of one axis.
Specify axis endpoint of ellipse or [Arc/Center]: *(Pick point 1.)*
3. Pick another point for the other end of the axis.
Specify other endpoint of axis: *(Pick point 2.)*
4. Show the half-distance to the other axis.
Specify distance to other axis or [Rotation]: *(Pick point 3.)*



The “distance to other axis” does not need to be perpendicular to the first axis.

ADDITIONAL METHODS FOR DRAWING ELLIPSES

The **ELLIPSE** command contains options that provide additional methods for drawing ellipses:

- **Center** starts with the center point of the ellipse, followed by the axes.
- **Rotation** starts with the rotation angle about the major axis.
- **Arc** constructs an elliptical arc.

Center

The **Center** option allows you to pick the center of the ellipse, and then to specify distance from the center point to the endpoints of the axes.

Command: **ellipse**

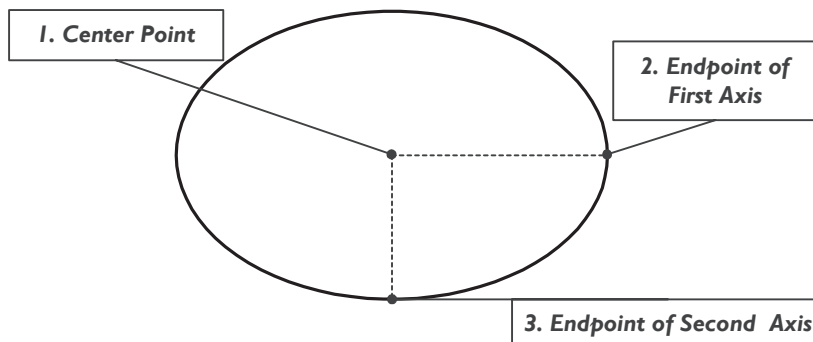
Specify axis endpoint of ellipse or [Arc/Center]: **c**

Specify center of ellipse: *(Pick point 1.)*

Specify endpoint of axis: *(Pick point 2.)*

Specify distance to other axis or [Rotation]: *(Pick point 3.)*

As you move the cursor during the “Specify distance to other axis” prompt, AutoCAD ghosts in the ellipse. Note that AutoCAD doesn’t care which axis you create first.



Rotation

The **Rotation** option rotates the ellipse about the major axis; it replaces the prompts for drawing the minor axis. (This option is also available when constructing ellipses with the **Center** option.) As you move the cursor during the “Specify rotation” prompt, AutoCAD ghosts the image of the ellipse.

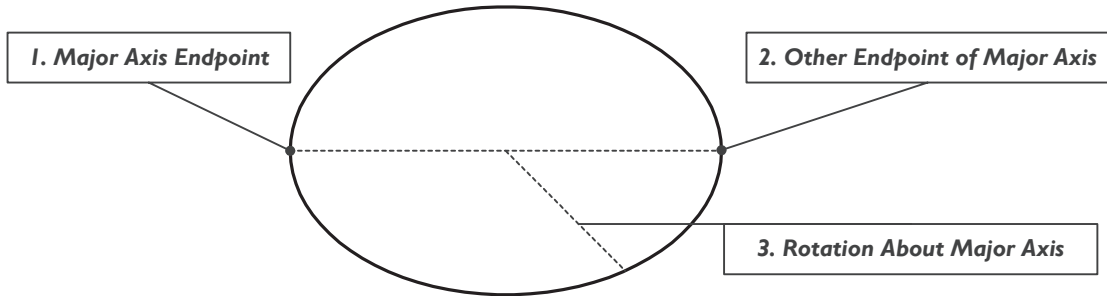
Command: **ellipse**

Specify axis endpoint of ellipse or [Arc/Center]: *(Pick point 1.)*

Specify other endpoint of axis: *(Pick point 2.)*

Specify distance to other axis or [Rotation]: **r**

Specify rotation around major axis: *(Pick point 3, or enter an angle.)*



Elliptical Arc

The **Arc** option draws elliptical arcs. The first few times you try drawing an elliptical arc, it might not turn out as desired. It takes some practice! Or, draw an ellipse and then trim or break it to an arc.

Command: **ellipse**

Specify axis endpoint of ellipse or [Arc/Center]: **a**

Specify axis endpoint of elliptical arc or [Center]: *(Pick point 1.)*

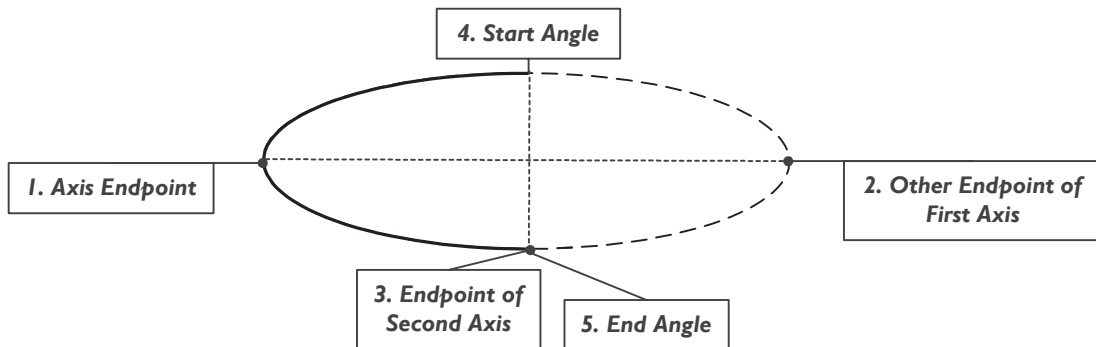
Specify other endpoint of axis: *(Pick point 2.)*

Specify distance to other axis or [Rotation]: *(Pick point 3.)*

Specify start angle or [Parameter]: *(Pick point 4.)*

Specify end angle or [Parameter/Included angle]: *(Pick point 5.)*

As you move the cursor during the “Start angle” and “End angle” prompts, AutoCAD ghosts a preview of the elliptical arc. The figure below shows an arc that consists of the upper half of an ellipse.



The **Arc** option has three methods of determining the arc: **Angle**, **Included Angle**, and **Parameter**.

The **Angle** option specifies the start angle and the end angle of the arc. The two angles are measured relative to the first axis endpoint. The *start angle* determines where the arc starts; the *end angle* where it ends. When you specify 0 degrees for the start angle, the arc starts at the first axis endpoint.

AutoCAD draws the arc in the counterclockwise direction. If you run into trouble, it's probably because you are trying to draw the arc clockwise.

The **Included Angle** option defines the arc by an angle that starts from the start angle (described above). The angle is relative to the start angle. AutoCAD prompts:

Specify end angle or [Parameter/Included angle]: **i**

Specify included angle for arc <180>: *(Enter an angle, or pick a point.)*

The **Parameter** option uses this formula to construct the elliptical arc :

$$p(u) = c + a \times \cos(u) + b \times \sin(u)$$

where:

a is the major axis.

b is the minor axis.

c is the center point of the ellipse.

Does anyone use this formula in the real world? Probably not. In any case, AutoCAD prompts you:

Specify start angle or [Parameter]: **p**

Specify start parameter or [Angle]: *(Pick a point.)*

Specify end parameter or [Angle/Included angle]: *(Pick a point.)*

Notice how you can switch back and forth between the three arc definitions – Angle, Included Angle, and Parameter.

Isocircle

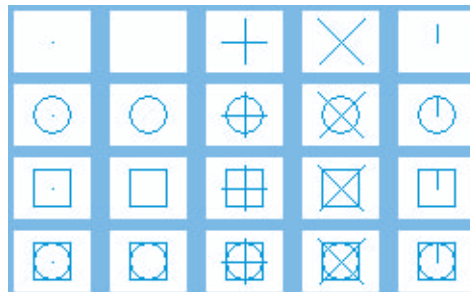
The **Isocircle** option appears in the **ELLIPSE** command's prompts only when isometric drafting mode is turned on with the **DSETTINGS** command.

POINT

The **POINT** command constructs *point* objects.

Points have no height or width: they are dots. Points are the smallest object that output devices can produce: single pixels on the screen, single dots on printed paper.

Because points are so small, they can be hard to see. Use the **PDMODE** and **PDSIZE** system variables to change the look and size of points. As illustrated below, points can be invisible, or combinations of lines, circles, and squares. The effect of the two system variables is retroactive, meaning changing their values changes the look and size of all previously-drawn points.



TUTORIAL: DRAWING POINTS

- I. To draw points, start the **POINT** command.
 - In the ribbon's 2D Home tab, choose the **Point** button in the Draw panel's flyout.

- Or, at the 'Command:' prompt, enter the **point** command:
Command: **point** (Press ENTER.)
 - Alternatively, enter the **po** alias at the 'Command:' prompt.
2. Notice that AutoCAD displays the values of the **PDMODE** and **PDSIZE** system variables.
Current point modes: PDMODE=0 PDSIZE=0.0000
 3. Pick a point to place the point.
Specify a point: (Pick a point.)

ADDITIONAL METHODS FOR DRAWING POINTS

The **POINT** command does not contain any options, but there are related commands that provide additional methods for drawing points:

- **MULTIPLE** command repeats the **POINT** command, as well as many other commands.
- **DDPTYPE** command changes the look of points.

Multiple

The **MULTIPLE** command repeats the **POINT** command, which is handy when you want to place more than one point at a time quickly. Indeed, the **MULTIPLE** command works with any command. At the command prompt, enter:

Command: **multiple**

Enter command name to repeat: **point**

Current point modes: PDMODE=0 PDSIZE=0.0000

Specify a point: (Pick a point.)

POINT Current point modes: PDMODE=0 PDSIZE=0.0000

Specify a point: (Press ESC or ENTER to exit repeating command.)

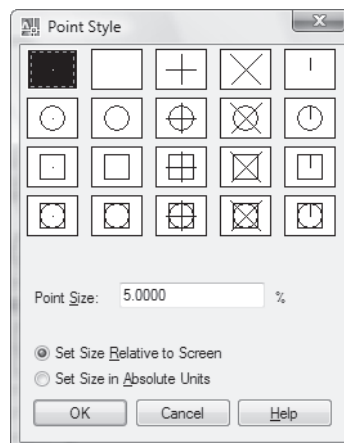
The command repeats until you press ESC or ENTER.

DdPType

The **DDPTYPE** command displays a dialog box that allows you to select the style and size of points. (DDPTYPE is short for "dynamic dialog point type.")

There are limitations: you can only choose from the styles listed in the dialog box, which excludes custom point styles; all points in the drawing take on the same style and size, which means different points cannot have different styles.

Enter **DDPTYPE** at the 'Command:' prompt to display the dialog box:



The upper half of the dialog box displays the point styles you can choose from. Note that the first style (the single dot) cannot be resized; the second style (the blank square) creates invisible points, which also cannot be sized.

The lower half changes the size of the point. The **Point Size** option sets the size as a percentage relative to the screen size or in units. The defaults are 5% and 5 units. I recommend you stick with the **Set Size Relative to Screen** (percentage) setting, because the **Absolute Units** setting can create points that are too large to see (when zoomed in) and too small to see (when zoomed out).

Click **OK** to accept the changes; AutoCAD automatically regenerates the drawing so that you can see the changes to the point display.

U AND UNDO

The **U** and **UNDO** commands reverse the effect of many (not all) commands.

- **U** — undoes the last action; this is the quick command to use when just one or two changes need to be undone.
- **UNDO** — provides many options for undoing the effects of commands; this is the advanced command that provides utter control.

You can draw, edit, and then undo your work — then redo it. This is useful if you have just performed an operation that you wish to reverse, either a mistake or a what-if scenario. After undoing one or more operations, you can use the **REDO** command *once* to reverse the undo, while **MREDO** redoes multiple undoes.

The **U** command reverses the effect of the most recent command. You can execute a series of **U** commands to back up through a string of changes. (The **U** command is not an alias for the more advanced **UNDO** command, although it does function identically to the **UNDO** command's **1** option.)

Undoing a command restores the drawing to the state before the command was executed. For example, erase an object, and then execute the **U** command: the object is restored. When you scale an object, and then undo it, the object is scaled back to its original size. Undoing a just-completed **BLOCK** command restores the block, and deletes the block definition that was created, leaving the drawing exactly as it was before the block was inserted.

At the command prompt, the **U** command lists the command that is undone to alert you to the type of command that was affected.

TUTORIAL: UNDOING COMMANDS

1. Open a drawing, and erase a part of it with the **ERASE** command.
2. To reverse the erasure, start the **U** command with one of these methods:
 - From the Edit menu, choose **Undo**.
 - On the Quick Action toolbar, choose **Undo**.
 - At the keyboard, press **CTRL+Z**.
 - At the 'Command:' prompt, enter the **u** command.

Command: **u** (Press ENTER.)

3. AutoCAD undoes the effect of the **ERASE** command, and then displays the name of the undone command:

ERASE

When you undo all the way back to the first command, AutoCAD reports:

Everything has been undone



Note Several commands cannot be undone. **SAVE**, **PLOT**, and **WBLOCK**, for example, are unaffected, because AutoCAD cannot “unsave” or “unplot.” If you attempt to use the **U** or **UNDO** command after these commands, the name of the command is displayed, but the command’s action is not undone.

(The way to undo a save is to retrieve the backup copy, as described later in this chapter. The way to undo a plot is to fold up the paper and then throw it in the recycling bin, suggests the tech editor.)

UNDO and **U** have no effect on commands that change the arrangement of windows (as opposed to viewports), and redraw or regenerate (such as **REGEN**) the drawing. The two undo commands also have no effect on commands that took place before the drawing was opened. For example, you work on a drawing, save it, and then close the drawing. When you open the drawing again, AutoCAD cannot undo the commands of the previous editing session.

UNDOING COMMANDS: ADDITIONAL METHODS

AutoCAD has several additional methods to undo the effects of commands:

- **UNDO** command controls the undo process at the command line.
- **Undo** and **Previous** options undo actions in commands; the **Cancel** button in dialog boxes.

Let’s look at each.

Undo

The **UNDO** command provides fine control over the undo process.

Command: **undo**

Enter the number of operations to undo or [Auto/Control/BEGIN/End/Mark/Back] <1>:
(Enter a number or an option.)

By default, the **UNDO** command operates like the **U** command: press **ENTER** at the “Enter the number of operations to undo” prompt, and AutoCAD undoes the last command (if possible). Enter a number, such as **4**, to undo the last four commands. This is the same as entering **U** four times.

Auto

The **Auto** option groups all the actions of a single command into a single undo. The **Auto** option is not available when the **Control** option is turned off.

Enter UNDO Auto mode [ON/OFF] <On>: (Enter **ON** or **OFF**.)

Control

The **Control** option limits the effect of the **UNDO** command:

Enter an UNDO control option [All/None/One] <All>: (Enter an option.)

The **All** option turns on the **UNDO** and **U** commands, and allows them to undo all the way to the beginning of the drawing session.

The **None** option turns off the **UNDO** and **U** commands, grays out the **Undo** button on the Standard toolbar, and discards the undo history. Use this option only if your computer is low on disk space.

The **One** option restricts the **UNDO** command to a single undo, and makes the **Auto**, **Begin**, and **Mark** options unavailable.

BEGIN/End

The **BEGIN** option groups several commands into a set.

Command: **undo**

Enter the number of operations to undo or [Auto/Control/BEGIN/End/Mark/Back] <1>: **be**

After you enter the **BEGIN** option, the **UNDO** command ends, but AutoCAD starts recording all the

commands you enter, until you enter the **End** option:

Command: **undo**

Enter the number of operations to undo or [Auto/Control/BEGIN/End/Mark/Back] <1>: **e**

The **UNDO**, **REDO**, and **U** commands now treat the set of commands as a single undo:

Command: **u**

GROUP

Mark/Back

The **Mark** option places a marker in the undo collection:

Command: **undo**

Enter the number of operations to undo or [Auto/Control/BEGIN/End/Mark/Back] <1>: **m**

The **Back** option undoes all commands back to the marker:

Enter the number of operations to undo or [Auto/Control/BEGIN/End/Mark/Back] <1>: **b**

Mark encountered

You may place as many marks as you require; the **Back** option moves back through the undo collection one mark at a time, removing each mark.



Note It is dangerous to use the **UNDO** command's **Back** option without marks, because it undoes every action in the current editing session:

Command: **undo**

Enter the number of operations to undo or [Auto/Control/BEGIN/End/Mark/Back] <1>: **b**

This will undo everything. OK? <Y> (Enter Y.)

Everything has been undone

Combined Zooms and Pans

When you perform several zooms and pans in a row, undoing them could be annoying if AutoCAD were to redraw each view change. Fortunately, AutoCAD groups all sequential view changes into a single undo through the **Combine zoom and pan commands** option. You find it in the Undo/Redo section of the User Preferences tab of the Options dialog box.

Undo, Previous Options and Cancel Buttons

Several commands include methods of undoing operations within them.

Undo Option

Commands that execute more than one action often include the **Undo** option. This allows you to reverse the effect of the last action without leaving the command. For example, the **LINE** command's **Undo** option erases the last-drawn segment. Other commands that have the undo option include **PLINE**, **PEDIT**, and **3DPOLY**.

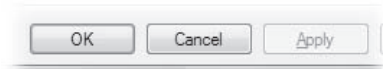
Previous Option

Instead of an undo option, other commands have an equivalent option called "previous." The **Zoom** command's **Previous** option, for example, displays the previous view, whether created by the **Zoom**, **Pan**, or **-SHADEMODE** command. This option remembers up to ten previous views.

Other commands with a **Previous** option include **SELECT** and **UCS**. One command is itself a "previous" command: **LAYERP** restores previous layer states.

Cancel Button

Many dialog boxes have a button labeled **Cancel**. Click this button after making changes to the dialog box, if you don't want to keep the changes.



REDO AND MREDO

The **REDO** and **MREDO** (short for “multiple redo”) commands are the antidotes to the **U** and **UNDO** commands: they reverse the undoing.

- **REDO** — redoes the last undo; this is the quick command to use when just one or two changes need to be reversed.
- **MREDO** — provides a couple of options for redoing the effects of undo; this advanced command is also available as a droplist on the toolbar.

The **REDO** command must be used immediately after one of the undo commands.

TUTORIAL: REDOING UNDOS

1. To undo an undo command, start the **REDO** command with one of these methods:
 - From the Edit menu, choose **Redo**.
 - At the keyboard, press **CTRL+Y**.
 - On the Quick Action toolbar, choose **Redo**.
 - At the ‘Command:’ prompt, enter the **redo** command.
 Command: **redo** (Press ENTER.)
2. AutoCAD redoes the previous undo, and then displays the name of the redone command:
 ARC

When you redo back to the first undo, AutoCAD reports:
 Everything has been redone

MRedo

The **MREDO** command provides additional control over the redo process.

Command: **mredo**

Enter number of actions or [All/Last]: (Enter a number or an option.)

Enter the number of undone commands you want reversed.

The **All** option reverses all previous undo actions.

The **Last** option reverses the last undo action only; this is like the **REDO** command.

CONSTRUCTING OTHER OBJECTS

This chapter has introduced you to AutoCAD's basic drawing commands. These allow you easily to draw lines, circles, rectangles, regular polygons, arcs, ellipses, elliptical arcs, polylines, and points. They don't, however, make it easy to draw other common 2D geometric shapes, such as isosceles and right triangles, spirals, parallelograms; as well as 3D objects, such as cubes and cylinders. You learn about 3D objects in chapter 18 of this book.

To draw other kinds of 2D shapes, for example the four kinds of triangle, you need to *construct* them, using one or more commands available in AutoCAD:

Equilateral — all three sides the same length.

Isosceles — two sides the same length.

Scalene — all three sides at different lengths.

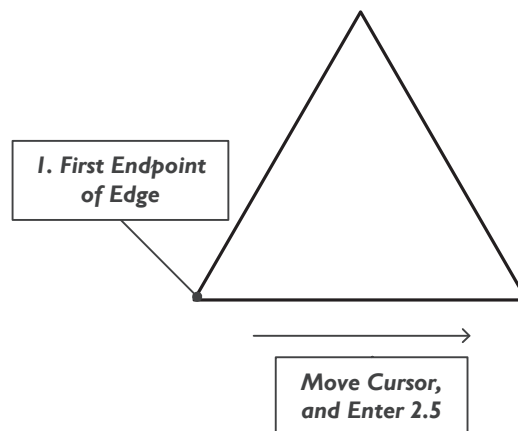
Right Angle — one angle at 90 degrees.

As you may recall from your high school geometry class, triangles are drawn with a combination of lengths and angles, and sometimes require the use of sine, cosine, and tangent calculations. Although it's not obvious, AutoCAD provides all the tools you need to draw any kind of triangle — without getting out the scientific calculator.

EQUILATERAL TRIANGLES

All three sides of equilateral triangles have the same length. (Again, if you recall your high school geometry class, the three interior angles are all 60 degrees.) This kind of triangle is most easily drawn with the **POLYGON** command set to draw 3 sides. When you know the length of the sides, use the **Edge** option. For example, if the sides are 2.5 units long, use *direct distance entry* to specify the length, as follows:

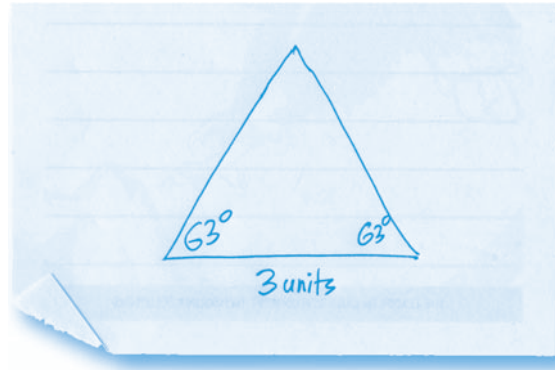
1. Start the **POLYGON** command:
Command: **polygon**
2. To draw triangles, specify 3 sides:
Enter number of sides <4>: **3**
3. By using the **Edge** option, you specify the length of one side; AutoCAD draws the other two sides to match.
Specify center of polygon or [Edge]: **e**
Specify first endpoint of edge: (*Pick point 1.*)
Specify second endpoint of edge: (*Move cursor in a direction, and then enter length.*) **2.5**



AutoCAD draws the equilateral triangle for you.

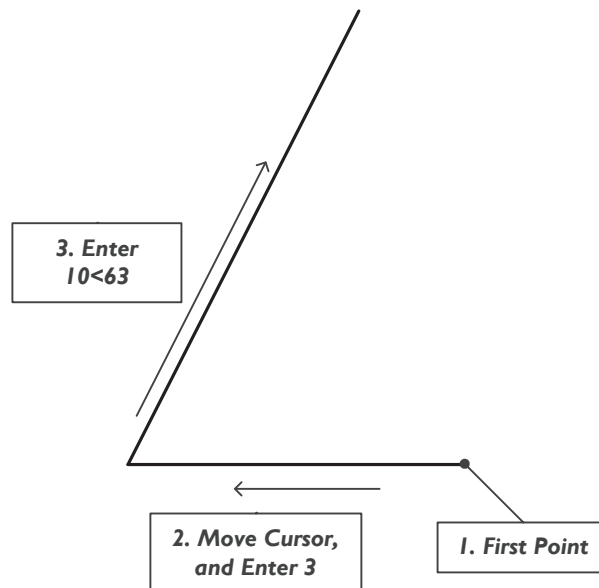
ISOSCELES TRIANGLES

Two sides of isosceles triangles have the same length (the two opposite angles also being the same). One method to draw this triangle is to use the **LINE**, **MIRROR**, and **TRIM** commands. For example, consider a triangle with a base of 3 units and isosceles angles of 63 degrees.



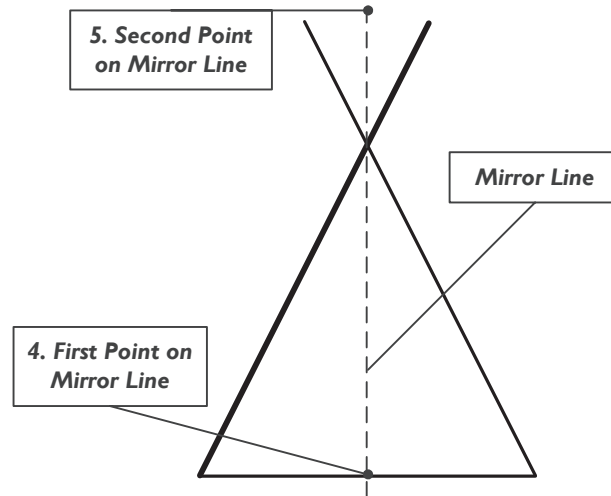
We don't know the length of the other two sides, so we draw them an arbitrary length, and then trim or extend to fit. For the best result, turn on polar mode.

1. Start the **LINE** command to draw the base and one angled leg:
 Command: **line**
 Specify first point: *(Pick point 1.)*
2. Use direct distance entry to draw the base 3 units long:
 Specify next point or [Undo]: *(Move cursor to the right, and then enter length.)* **3**
3. Use relative coordinates to draw one leg 10 units long at 63 degrees. The 10 units is arbitrary, because we do not know its length; we edit the length later.
 Specify next point or [Undo]: **10<63**
 Specify next point or [Close/Undo]: *(Press ENTER to exit command.)*

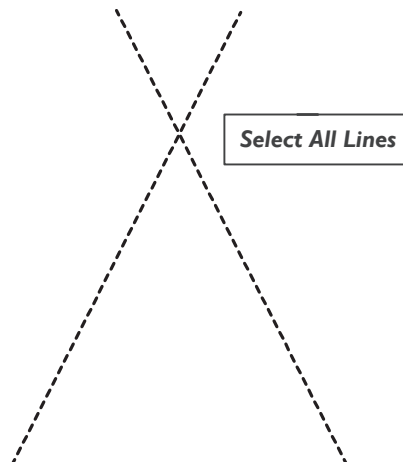


4. Start the **MIRROR** command to make a mirror copy of the angled leg:
 Command: **mirror**

5. Enter **L** to select the last-drawn object:
 Select objects: **L**
 1 found Select objects: *(Press ENTER to end object selection.)*
6. Use MIDpoint object snap to create an imaginary vertical mirroring line:
 Specify first point of mirror line: **mid**
 of *(Pick point 4.)*
 Specify second point of mirror line: *(Pick point 5.)*



7. Keep the source object, and exit the command:
 Delete source objects? [Yes/No] <N>: *(Press ENTER to accept default of **No**, and then exits the command.)*
8. Use the **TRIM** command to trim back the two angled lines to their intersection point:
 Command: **trim**
 Current settings: Projection=UCS, Edge=None
 Select cutting edges ...
(Filleting with radius = 0 also works.)
9. Select all the lines. They become each other's cutting edge:
 Select objects or <select all>: *(Press ENTER to select all objects.)*



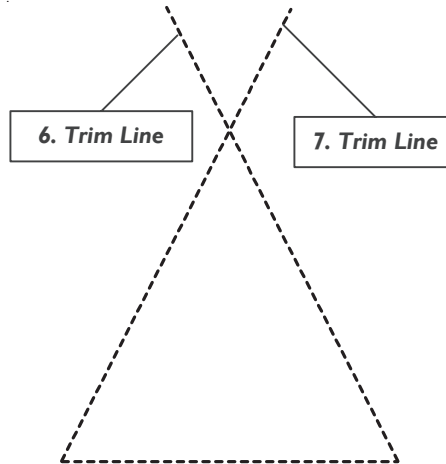
10. Select the two line extensions to trim them. Be sure to pick the portion of the lines to be trimmed away:

Select object to trim or shift-select to extend or

[Fence/Crossing/Project/Edge/eRase/Undo]: *(Select end of one angled line, 6.)*

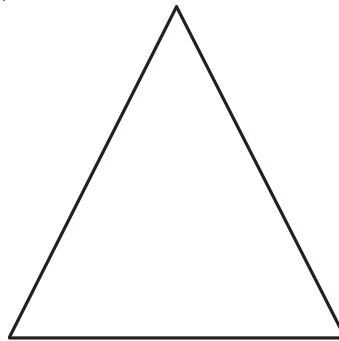
Select object to trim or shift-select to extend or

[Fence/Crossing/Project/Edge/eRase/Undo]: *(Select end of the other line, 7.)*



Select object to trim or shift-select to extend or

[Fence/Crossing/Project/Edge/eRase/Undo]: *(Press ENTER to exit command.)*

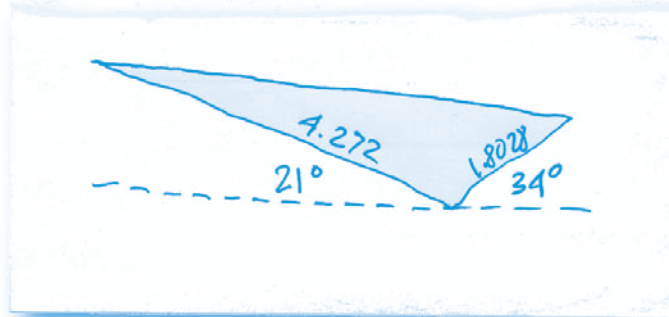


The isosceles triangle is complete. The technical editor found another method that requires just one application of the **LINE** command:

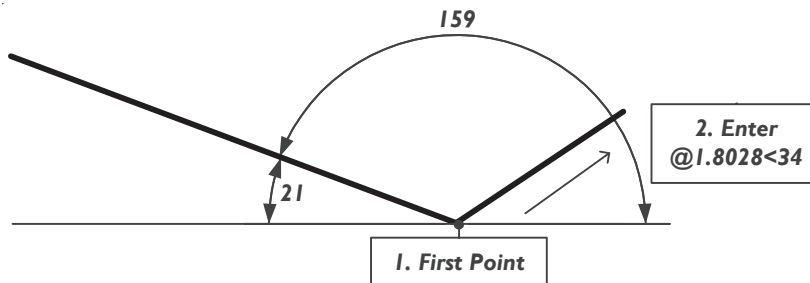
1. Set running object snap to **MIDpoint**. Turn on object tracking.
2. Start the **LINE** command to draw the base and one angled leg:
 Command: **line**
 Specify first point: *(Pick point 1.)*
3. Use direct distance entry to draw the base 3 units long:
 Specify next point or [Undo]: *(Move cursor to the right, and then enter length.)* **3**
4. Draw one leg any length at 63 degrees.
 Specify next point or [Undo]: **<63**
5. Pick up the tracking point at the midpoint of the first line.
6. Track to and select the upper vertex.
7. Enter **c** to close the polygon.

SCALENE TRIANGLES

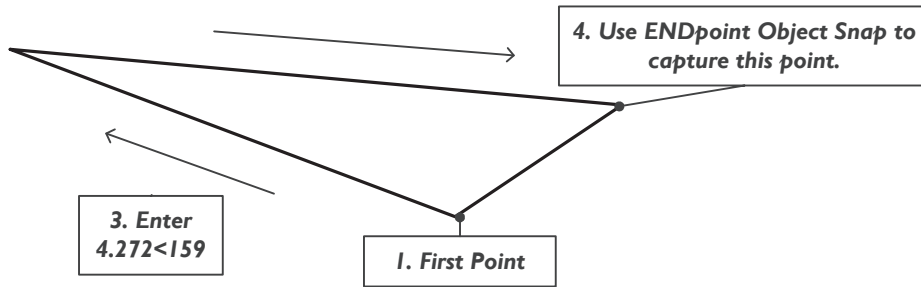
All three sides and corners of scalene triangles have different lengths and angles. This kind of triangle is best drawn with the **LINE** command, using a combination of direct distance entry and polar coordinates, as required. Take, for example, a triangle with the following specifications:



1. Start the **LINE** command:
 Command: **line**
 Specify first point: *(Pick point 1).*
2. Use polar coordinates (which are also relative) to draw the short leg at right:
 Specify next point or [Undo]: **1.8028<34**
3. End the **LINE** command, and start over to draw the next leg (at left):
 Specify next point or [Undo]: *(Press ENTER.)*
 Command: *(Press ENTER.)*
 LINE Specify first point: **endp**
 of *(Again, pick point 1).*



4. Use polar coordinates to draw the other leg. The angle of 159 degrees = $180 - 21 = 159$.
 Specify next point or [Undo]: **4.272<159**
5. Use **ENDpoint** object snap to draw the third leg:
 Specify next point or [Undo]: **end**
 of *(Pick point 4.)*
 Specify next point or [Close/Undo]: *(Press ENTER.)*

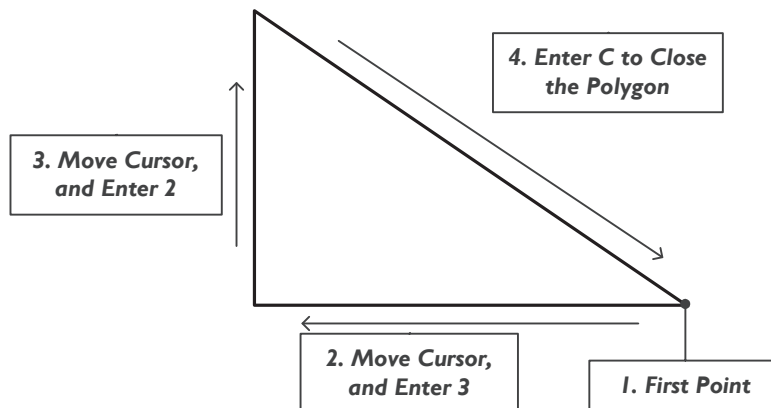


RIGHT ANGLE TRIANGLES

One corner of right angle triangles is at 90 degrees; the two remaining corners have other angle that add up to 90 degrees, such as the same angle (45 degrees each, making them isosceles triangles), or two different angles, making them scalene.

This kind of triangle is easily drawn with the **LINE** command. When you know the length of two sides, such as 3 and 2 units, follow these steps:

1. Turning on ortho mode helps you draw the 90-degree angle easily:
(Press **F8** to turn on ortho mode.)
2. Start the **LINE** command:
Command: **line**
Specify first point: (Pick point 1.)
3. Notice that it is easier to draw this kind of triangle when you start at a corner away from the right angle, and then draw toward the right angle.
Specify next point or [Undo]: (Move cursor to the left, and then enter length.) **3**
Specify next point or [Undo]: (Move cursor up, and then enter length.) **2**
4. By using the **Close** option, you don't have to work out the length of angle of the third side.
Specify next point or [Close/Undo]: **c**



The right angle triangle is complete.

EXERCISES

1. With the **LINE** command, draw the house shape shown below:

Command: **line**

Specify first point: *(Enter point 1.)*

Specify next point: *(Enter point 2.)*

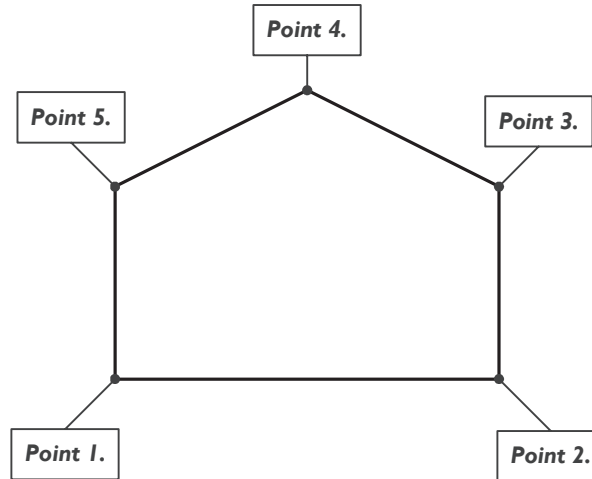
Specify next point: *(Enter point 3.)*

Specify next point: *(Enter point 4.)*

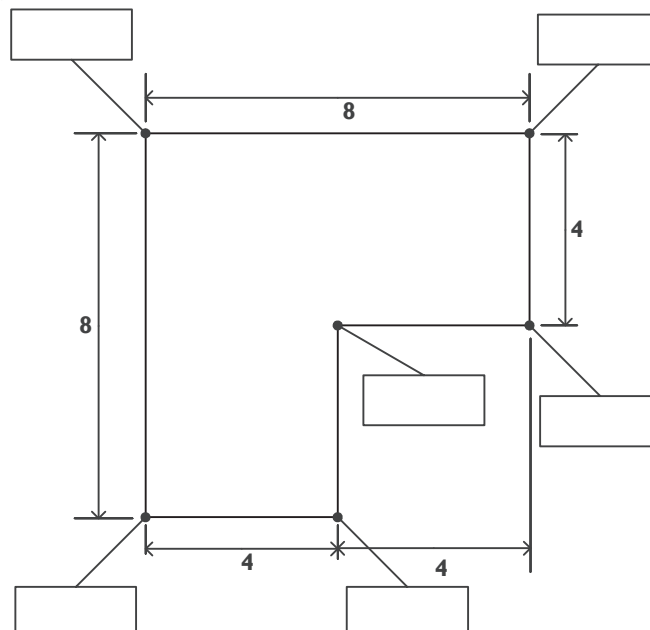
Specify next point: *(Enter point 5.)*

Specify next point: *(Enter point 1 again.)*

Notice that the line stretched behind the crosshairs. This is called “rubber banding.”



2. Using the figure below, fill in the missing absolute coordinates. Each side of the floor plan is dimensioned. Place the answers in the boxes provided. Assume the origin is at the lower left corner.



3. Using the **LINE** command, connect the following points designated by absolute coordinates.

Point 1: **1,1**

Point 2: **5,1**

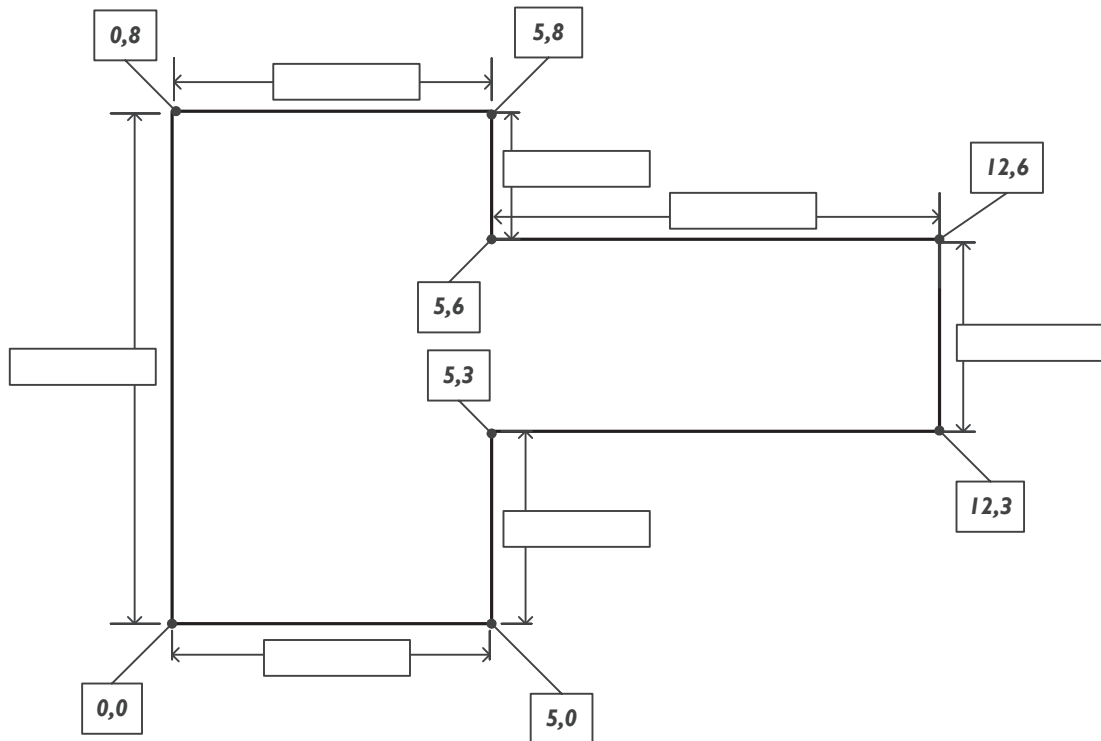
Point 3: **5,5**

Point 4: **1,5**

Point 5: **1,1**

What shape did you draw?

4. Determine the length of each side of the floor plan in the figure below. (Calculate the lengths from the absolute coordinates given.)



5. Use the following absolute and relative coordinates to draw an object.

Point 1: **0,0**

Point 2: **@3,0**

Point 3: **@0,1**

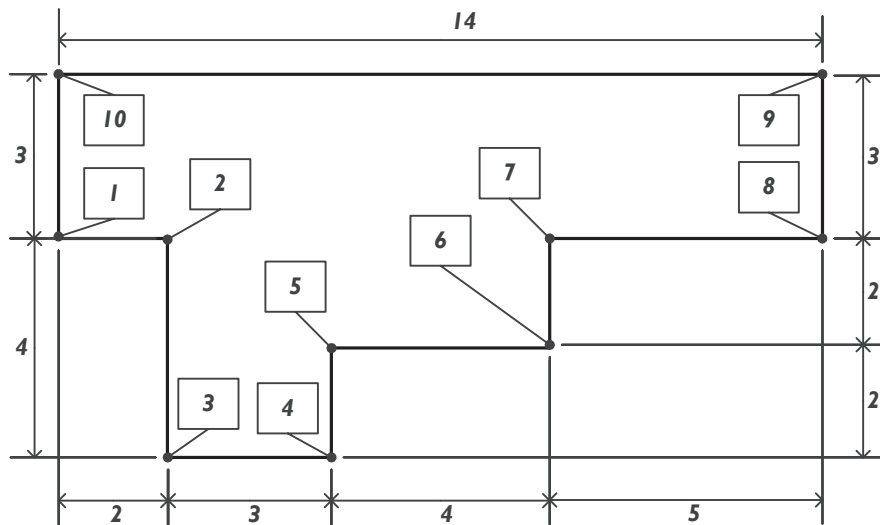
Point 4: **@-2,0**

Point 5: **@0,2**

Point 6: **@-1,0**

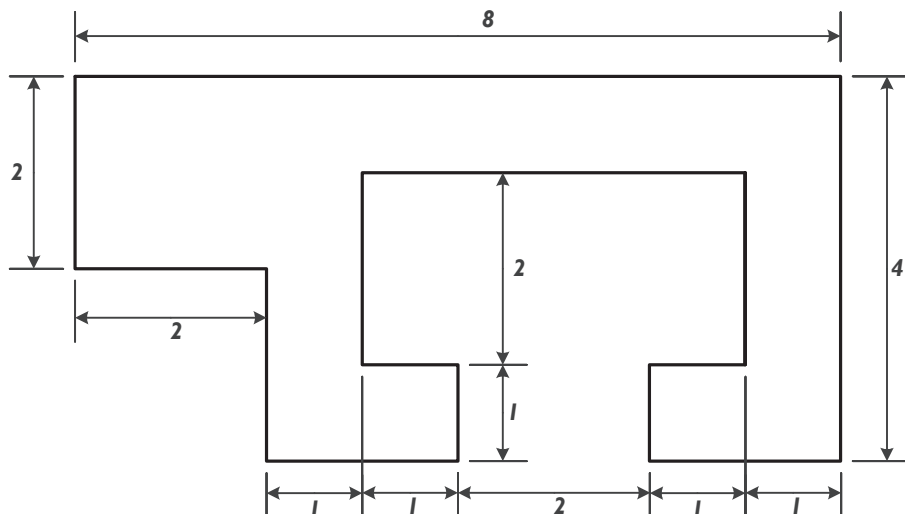
Point 7: **0,0**

6. List the relative coordinates used to draw the following baseplate.



1. _____, _____
2. _____, _____
3. _____, _____
4. _____, _____
5. _____, _____
6. _____, _____
7. _____, _____
8. _____, _____
9. _____, _____
10. _____, _____

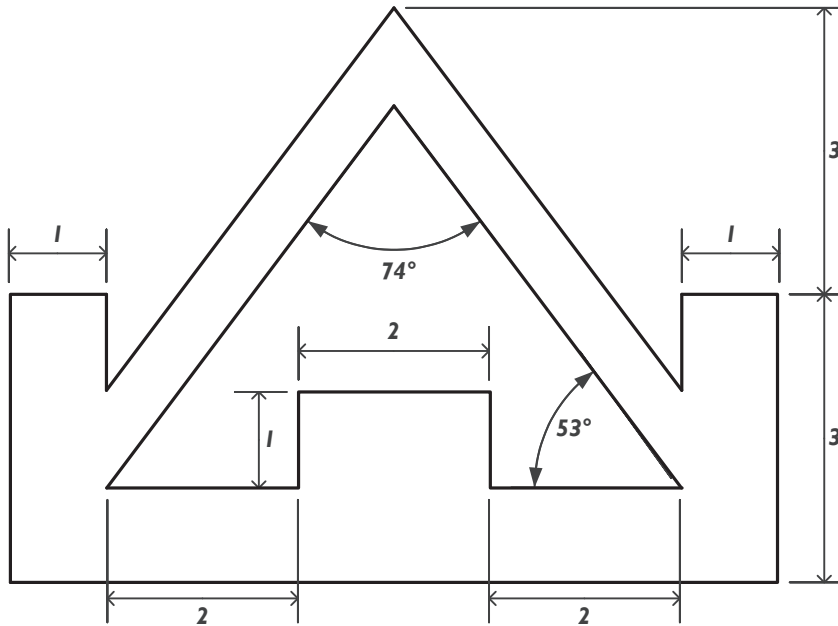
7. List the polar coordinates used to draw the baseplate above.
8. Write a list of the absolute coordinates used to construct the shim shown in the figure below. The origin is at the lower left corner.



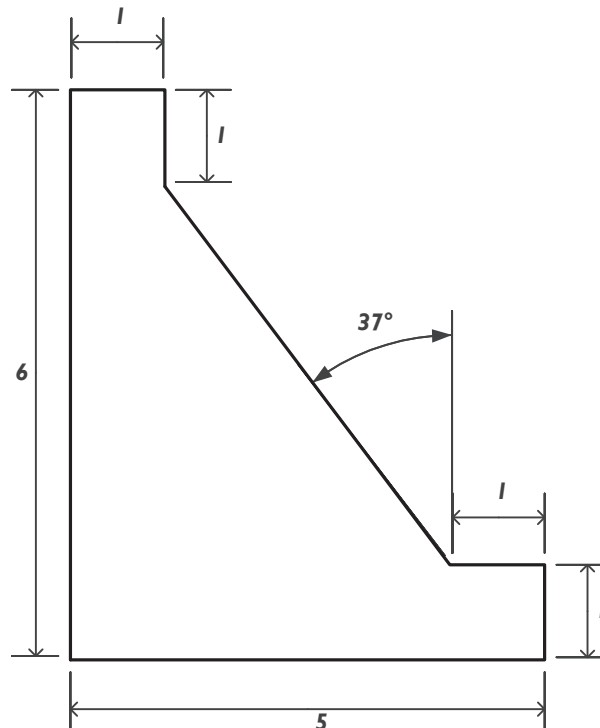
9. Use the following relative distances and angles to draw an object.

Point 1: 0,0
 Point 2: @4<0
 Point 3: @4<120
 Point 4: @4<240

10. Write a list of the relative coordinates used to construct the support plate shown in the figure below. The origin is at the lower left corner.



11. Write a list of the polar coordinates used to construct the corner shelf shown in the figure below.



12. Set up a point type of 66, and then place it in the drawing:

Command: **pdmode**

New value for PDMODE <default>: **66**

Recall that **PDMODE** affects all points in the drawing. Points placed previously are updated with the next command that causes a regeneration, such as **REGEN**. To place the point in the drawing, enter:

Command: **point**

Current point modes: PDMODE=66 PDSIZE=0.0000

Specify a point: *(Pick a point.)*

13. Construct a circle with a *radius* of 5. From the Home tab's **Draw** panel, choose **Circle | Center, Radius**.

Command: **_circle**

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: *(Pick a point.)*

Specify radius of circle or [Diameter]: **5**

14. Construct a circle using a center point and a *diameter* of 3.

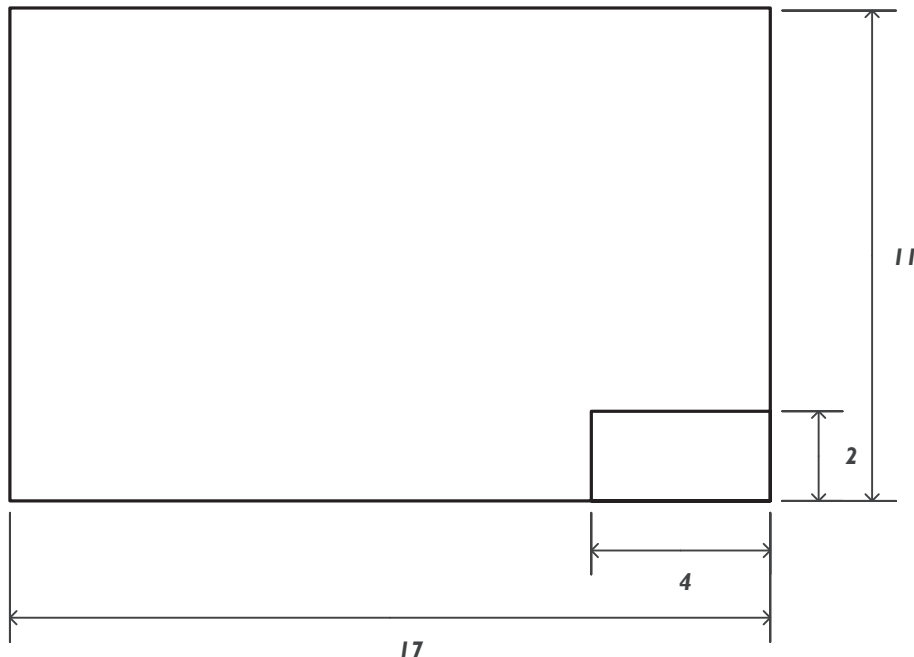
Command: **circle**

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: *(Enter point 1.)*

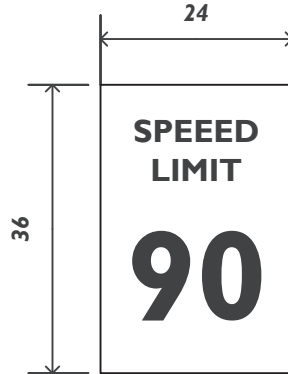
Specify radius of circle or [Diameter]: **d**

Specify diameter of circle <5.0000>: **3**

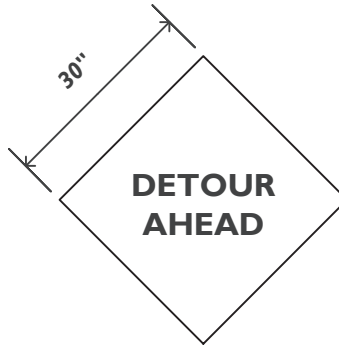
15. With the **RECTANG** command, draw two rectangles. One rectangle represents a B-size drawing sheet (17" x 11"); the second represents the title block in the lower right corner (4" x 2").



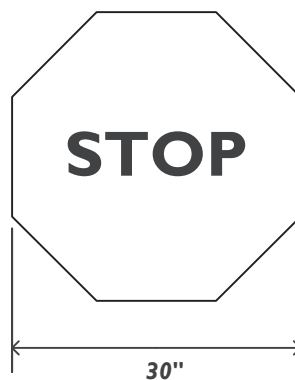
16. With the **RECTANGLE** command, draw the outline of a standard 24" x 36" speed limit sign; do not draw the text.



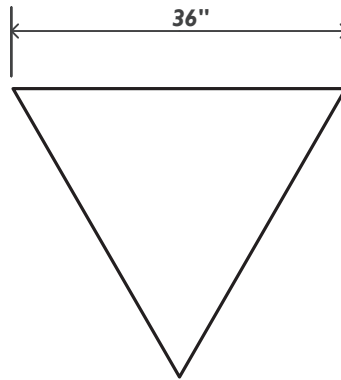
17. Use the **POLYGON** command to draw the outline of a standard 30" warning sign; do not draw the text:



18. Use the **POLYGON** command to draw the outline of a standard 30" Stop sign; do not draw the text:



19. With the **POLYGON** command, draw the outline of a standard 36" Yield sign:

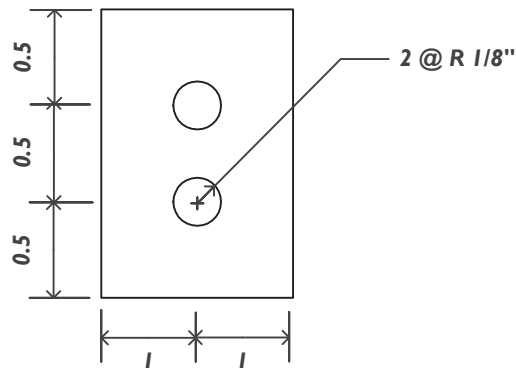


20. Draw donuts with the following diameters:

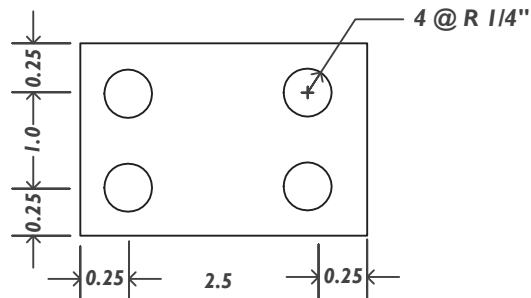
- a. ID = 1.0; OD = 1.5
- b. ID = OD = 2.5
- c. ID = 0.0; OD = 1.0

21. Use drawing commands discussed in this chapter to draw the following baseplates:

a. Rectangular baseplate with two holes:

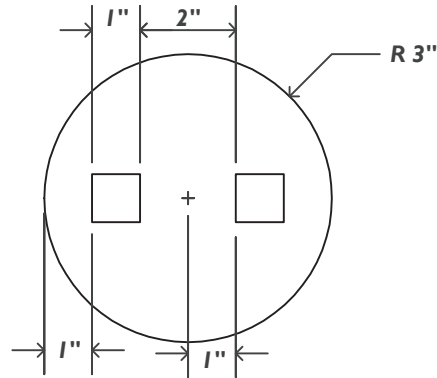


b. Rectangular baseplate with four holes:

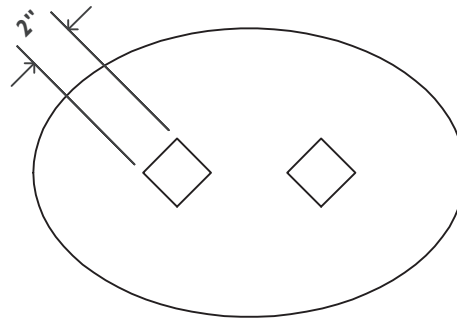


22. Use drawing commands discussed in this chapter to draw the following baseplates:

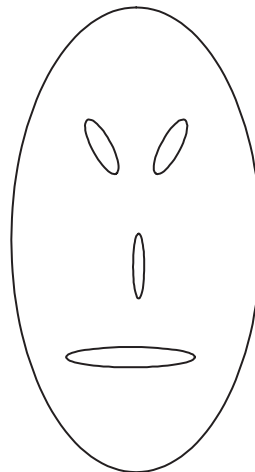
a. Circular baseplate with two square holes:



b. Elliptical baseplate with two square holes. The minor radius is 3", and the major radius is 4.5":



23. Use the **ELLIPSE** command to draw the face of an alien. Use the figure below to help you. Don't worry about the size.



24. Draw a compact disc, which has an outer diameter of 4.75" and an inner diameter of 0.6". What are the equivalent measurements as radii?

25. Draw arcs with the following specifications:

- a. Start point **7,5**
Second point **9.5,3.5**
Endpoint **6,2**
- b. Center point **4,4**
Start point **4,2**
Angle **270**
- c. Start point **10.5,5.5**
Center point **9,6**
Length **3**

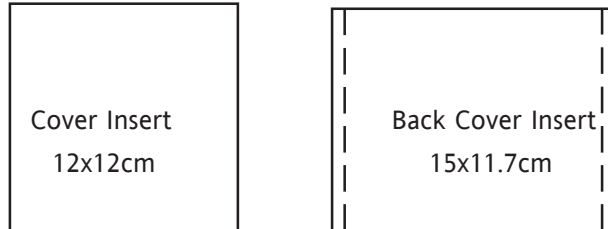
26. Draw circles with the following specifications:

- a. Center point **5,5**
Radius **3.5**
- b. Center point **5,5**
Diameter **3.5**
- c. 2P: first point **10,5**
Second point **3,2**
- d. 3P: first point **4,7**
Second point **6,5**
Third point **5,2**

27. Draw the 4.5" x 2.75" electrical cover plate shown below. The screw holes are 1/8" in diameter, and 2-5/16" apart. The light switch cover plate has an opening of 0.95" x 0.4".



28. With the **Line** and **Arc** options of the **PLINE** command, draw the outline of a standard credit card — 3.35" wide by 2.15" high, with a 0.1"-radius arc at each corner.
29. Draw the inserts for CD cases:
- The cover insert is 12cm x 12cm.
 - The back cover insert is 15cm wide x 11.7 cm tall; the two spine strips are 6.4 mm wide.



30. Draw the bicycle shown in the figure below.
Use polylines for the frame, lines for the spokes, and donuts for the tires.



CHAPTER REVIEW

1. What is another name for a single line?
2. Is using the **Close** option of the **LINE** command more accurate at closing the polygon than attempting to line up endpoints manually?
3. What does the @ symbol mean?
The < symbol?
4. How do you continue drawing a line *tangent* to a previously-drawn arc?
5. Name the best command for drawing the following triangles:
Equilateral triangle
Scalene triangle
6. What is AutoCAD's default method of constructing arcs?
7. What command can be used to change the look and size of points?
8. When rectangles are drawn with the **RECTANG** command, can each corner have a different fillet radius?
9. The **POLYGON** command uses three methods to determine the size of polygons. Explain the meaning of the methods:
Edge
Inscribed
Circumscribed
10. The **PDMODE** and **PDSIZE** system variables are *retroactive*. What does this mean?
11. There are six methods of constructing circles. What four circle properties are used in various combinations to comprise these methods?
a.
b.
c.
d.
12. Describe the meaning of these options for drawing circles:
2P
3P
Ttr
13. Describe a method of drawing arcs that does not involve the **ARC** command.
14. List three methods for ending the **LINE** command?
a.
b.
c.
15. In which direction does AutoCAD draw arcs (by default)?
16. What is the difference between a *minor* arc and a *major* arc?
17. What is the *chord* of an arc?
18. How do you continue drawing an arc tangent to a previously-drawn arc or line?
19. What does the **ARC** command's **Direction** option specify?
20. What must you do to end the **DONUT** command?
21. What is the alias for the following commands:
ARC
LINE
CIRCLE
PLINE

22. Where on the ribbon do you find commands for drawing objects?
23. Name the three kinds of circles drawn by the **DONUT** command:
 - a.
 - b.
 - c.
24. What do the **POLYLINE**, **DONUT**, and **POLYGON** commands have in common?
25. Can polylines have varying width?
26. What is the purpose of the **PLINE** command's **Arc** option?
27. When selecting a wide polyline, where must you pick it?
28. Describe the purpose of the **PLINE** command's **Length** option.
29. What is the difference between the *minor* axis and the *major* axis of an ellipse?
30. What is another name for an ellipse with a rotation of 0 degrees?
31. Which must you specify first, the minor axis or the major axis of an ellipse?
32. Describe the purpose of the **MULTIPLE** command.
33. List two methods of drawing circles without using the **CIRCLE** command:
 - a.
 - b.
34. What is the effect of chamfers and fillets on the **RECTANGLE** command's **Area** option?
35. Does the **POLYGON** command draw scalene triangles?
36. Does the **RECTANGLE** command draw rhomboids?

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Drawing with Precision

One major advantage of CAD over hand drafting is precision. Instead of drawing with an accuracy to the nearest pencil (or pen) width, AutoCAD draws with an accuracy to 14 digits*. In this chapter, you learn to use some of AutoCAD's commands for drawing with precision, and to view the drawing in different ways. They are:

GRID displays a grid of dots or lines.

DSETTINGS controls drafting settings.

SNAP specifies the cursor increment and additional drafting options.

ORTHO constrains cursor movement to the horizontal and vertical.

POLAR helps draw at specific angles.

OSNAP helps accurately select geometric features.

OTRACK implements object snap tracking.

RAY and **XLINE** place construction lines in the drawing.

REDRAW and **REGEN** clean up and update the drawing.

ZOOM enlarges and reduces the view of the drawing.

PAN moves the view around.

CLEANSCREENON minimizes the user interface to maximize the drawing.

VIEW stores and recalls views of drawings by name, and controls viewport backgrounds.

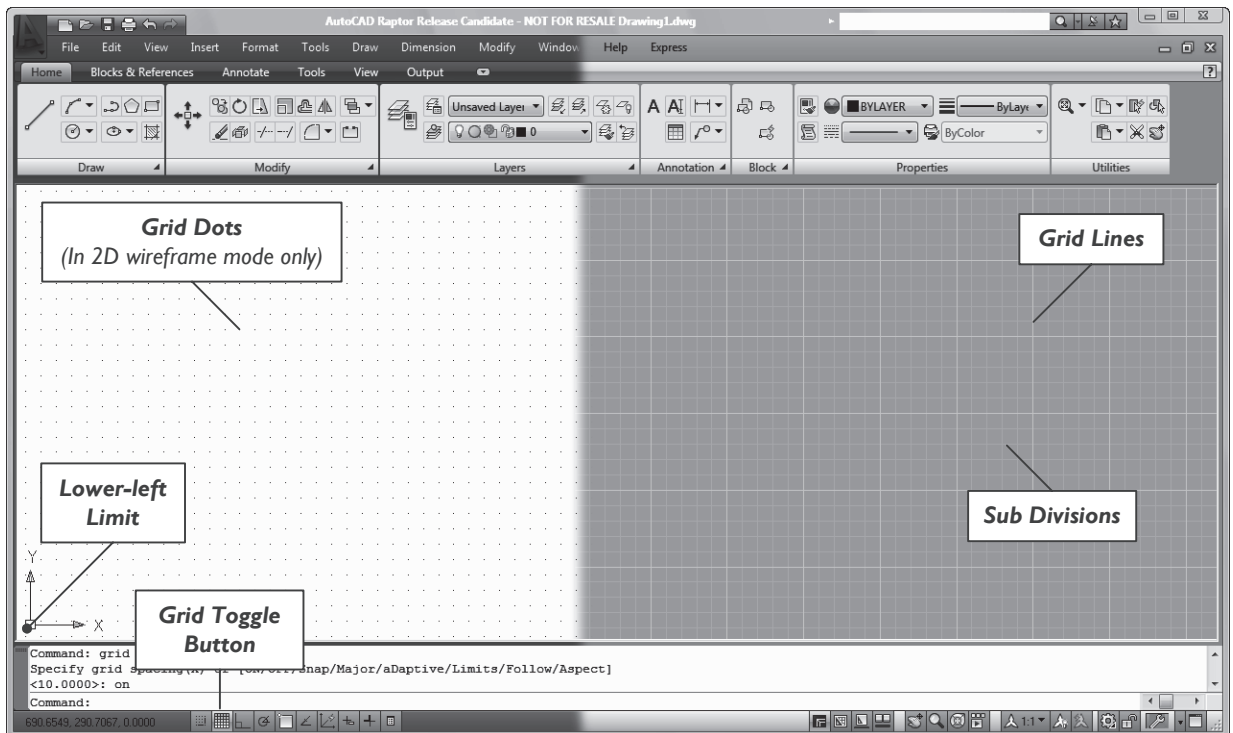


GRID

The **GRID** command toggles (turns on and off) the *grid*, a display of dots or lines. The grid is like a sheet of graph paper that helps you see the horizontal and vertical frame of reference. In 2D wireframe mode, AutoCAD displays dots (the intersections of the lines); in other modes, it displays a grid of lines.

The grid gives you a feeling for the size of the drawing and for distances within the drawing. Because it is a visual aid, the grid is not plotted.

You can change the spacing of grid dots and lines, the subspacing between grid lines, the angle of the grid, the style (standard or isometric), the extent of the grid, and the color of grid lines.



Left: Grid dots displayed in 2D wireframe mode.

Right: Grid lines displayed in all other modes.

TUTORIAL: TOGGING THE GRID

1. To turn on the grid, use the **GRID** command by one of these methods:

- From the status bar, choose **GRID** or  (turned on when button looks blue).
- On the keyboard, press function key **F7** or **CTRL+G**.
- Or, at the 'Command:' prompt, enter the **grid** command:

Command: **grid** (Press ENTER.)

2. Use the **ON** option to turn on the grid:

Specify grid spacing(X) or [ON/OFF/Snap/Major/aDaptive/Limits/Follow/Aspect] <0.5>: **on**

Notice that the grid is displayed.

3. To turn off the grid, repeat the command with the **OFF** option:

Command: **grid** (Press ENTER.)

Specify grid spacing(X) or [ON/OFF/Snap/Major/aDaptive/Limits/Follow/Aspect] <0.5>: **off**

CONTROLLING THE GRID DISPLAY: ADDITIONAL METHODS

When you start new drawings in AutoCAD, the grid spacing is 0.5 units. You can increase and decrease the spacing, make the spacing the same as the snap distance (as detailed later in this chapter), and change the spacing in the x and y directions separately.

To effect these changes, you can use a dialog box or the command line:

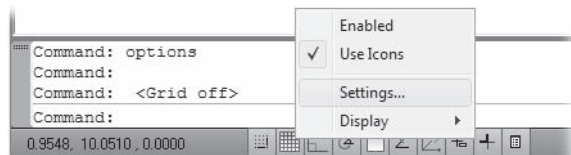
- **DSETTINGS** displays a dialog box for changing grid settings.
- **GRID** specifies the grid spacing and aspect ratio via the command line.
- **LIMITS** controls the extent of the grid.

Let's look at how each command affects the grid display.

DSettings Command

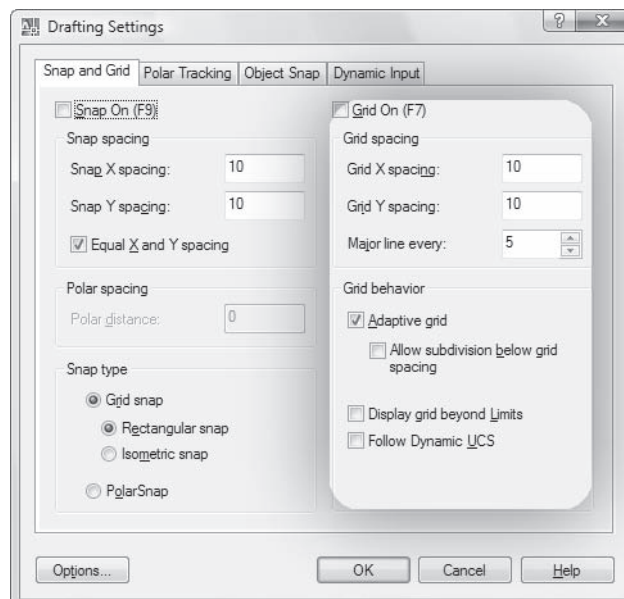
The **DSETTINGS** command (short for “drafting settings”) displays a dialog box with settings that control many (but not all) aspects of the grid display. The dialog box lists some of the options found in the **GRID** and **SNAP** commands.

- To access the Drafting Settings dialog box:
 - The easiest way is to right-click **GRID** (or ) on the status bar. From the shortcut menu, select **Settings**.

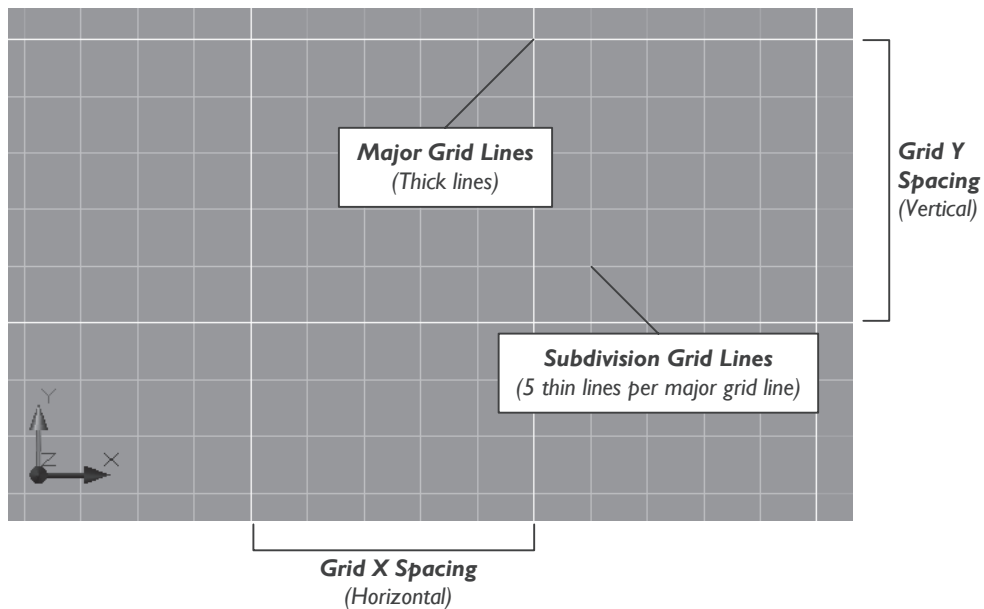


- At the ‘Command:’ prompt, enter the **dsettings** command.
Command: **dsettings**
- Or, enter one of the aliases — **ds**, **se** (short for “settings”), or **ddrmodes** (the old name for this command) at the ‘Command:’ prompt.

Notice that AutoCAD displays the Drafting Settings dialog box. (If necessary, choose the **Snap and Grid** tab.)



2. In this dialog box, you can make the following changes to the grid settings:
 - **Grid On (F7)** toggles the grid display. The “(F7)” notation reminds you that you can also press function key **F7** to turn the grid on and off.
 - **Grid X spacing** specifies the horizontal spacing between grid dots or lines. When you enter a value for the x spacing, AutoCAD automatically changes the y-spacing to match, if the **Equal X and Y spacing** option is turned on (a check mark shows).
 - **Grid Y spacing** specifies the vertical spacing between dots or lines. Some disciplines, such as highway design, use a different scale in the x-direction than in the y-direction. To make the vertical spacing different from the horizontal, you must turn off the **Equal X and Y spacing** option (under Snap spacing).
 - **Major line every** specifies the number of subdivision lines per major line; subdivisions are not seen in 2D wireframe mode.



- **Adaptive grid** makes the grid spacing wider when zoomed out. (AutoCAD no longer complains that the grid is too dense to display.)
 - **Allow subdivision below grid spacing** displays more and more minor grid lines as you zoom into the drawing.
 - **Display grid beyond limits** allows the grid display to go beyond the limits set by the **LIMITS** command.
 - **Follow Dynamic UCS** forces the grid to match the x,y-orientation of the dynamic UCS, when activated.
3. To accept the changes, click **OK**.

Grid Spacing

The **GRID** command lets you change the spacing between grid lines:

To make the grid spacing the same as the snap distance, use the **Snap** option:

Command: **grid** (Press **ENTER**.)

Specify grid spacing(X) or [ON/OFF/Snap/Major/aDaptive/Limits/Follow/Aspect] <0.5>: **s**

To set the grid spacing as a multiple of the snap distance, add an “x” to the **GRID** command’s default option.

For example, to make the grid spacing twice that of the snap spacing, enter **2x**, as follows:

Command: **grid** (Press ENTER.)

Specify grid spacing(X) or [ON/OFF/Snap/Major/aDaptive/Limits/Follow/Aspect] <0.5>: **2x**

LIMITS

The grid could extend nearly infinitely in all directions, and so the extent of the grid is limited by the **LIMITS** command. Typically, you want the grid covering the area in which you are drafting. Often, the lower-left corner is kept at 0,0 and only the location of the upper-right corner is changed, as follows:

Command: **limits**

Reset Model space limits:

Specify lower left corner or [ON/OFF] <0.0000,0.0000>: (Press ENTER, or enter x, y coordinates.)

Specify upper right corner <12.0000,9.0000>: (Enter x,y coordinates.)



Notes The grid can be set independently in each viewport, layout, and drawing.

Grid lines are colored light gray, but you can change the color in the **Display** tab of the **OPTIONS** dialog box: click the **Colors** button. The grid lines that lie on the x and y axes are colored differently: red for x and green for y, which matches the color coding of the UCS icon.



SNAP

The **SNAP** command specifies the cursor increment.

For example, if you are drafting a drawing to the nearest one inch, it's useful to set the snap to 1". When turned on, snap restricts the cursor to the interval you specify — 1" in this example. The cursor seems to jump from point to point. Snap is often used in conjunction with the grid.

As with the grid, you can change the snap's spacing, aspect ratio, and so on. Similarly, the snap can be set individually for each viewport, layout, and drawing. (The **LIMITS** command does not apply to snap.)

TUTORIAL: TOGGING SNAP MODE

1. To turn on snap, invoke the **SNAP** command by one of these methods:

- On the status bar, click **SNAP** or  (turned on when button looks blue).
- On the keyboard, press function key **F9** or **CTRL+B**.
- Or, at the 'Command:' prompt, enter the **snap** command.

Command: **snap** (Press ENTER.)

- Alternatively, enter the **sn** alias at the 'Command:' prompt.

2. Use the **ON** option to turn on snap:

Specify snap spacing or [ON/OFF/Aspect/Rotate/Style/Type] <0.5000>: **on**

When you move the mouse, notice that the cursor jumps.

3. To turn off snap, repeat the command with the **OFF** option:

Command: **grid** (Press ENTER.)

Specify snap spacing or [ON/OFF/Aspect/Rotate/Style/Type] <0.5000>: **off**

Notice that the cursor no longer jumps.

CONTROLLING THE SNAP: ADDITIONAL METHODS

When you start new drawings in AutoCAD, the default snap spacing is set to 0.5 units — the same as the grid. You can increase and decrease the spacing, make the spacing the same as the grid distance, or specify different spacing in the x and y directions. To make these changes, you can use a dialog box, or enter the changes at the command line, using these commands:

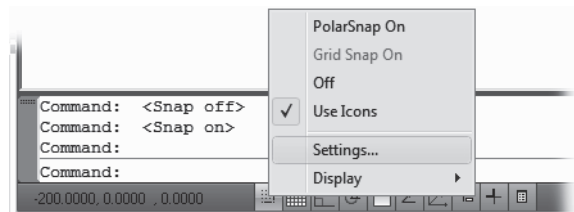
- **DSETTINGS** displays a dialog box for changing snap settings.
- **SNAP** specifies the snap spacing, aspect ratio, and modes via the command line.

Let's look at how each command affects the snap.

DSettings

The **DSETTINGS** command displays a dialog box that controls all settings affecting the snap. (The **Snap and Grid** tab combines the **SNAP** and **GRID** commands.)

- To access the Drafting Settings dialog box, right-click **SNAP** on the status line. From the shortcut menu, select **Settings**.



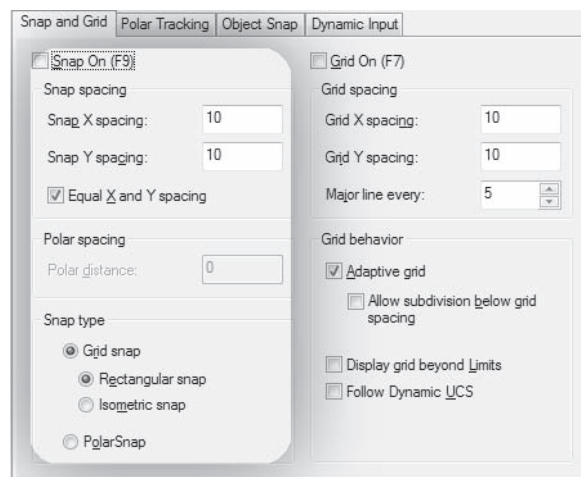
Note The shortcut menu (shown above) has settings that you can access without going through the dialog box:

Polar Snap On turns on polar snap mode, where the cursor aligns with distances and angles.

Grid Snap On turns on both the snap and the grid.

Off turns off snap mode.

AutoCAD displays the Drafting Settings dialog box, showing the **Snap and Grid** tab.



- In this dialog box, you can make the following changes to the snap settings:
 - **Snap On (F9)** toggles the snap on and off. The “(F9)” reminds you that you can press **F9** to turn snap mode on and off.
 - **Snap X spacing** specifies the horizontal snap increment. When you enter a value for the x-spacing, AutoCAD automatically changes the y-spacing to match, if **Equal X and Y spacing** is turned on. (The old press-the-TAB-key trick no longer works.)

- **Snap Y spacing** specifies the vertical snap increment.
 - **Grid Snap** sets the snap spacing equal to the grid spacing, and ignores the values set earlier in the **Snap X and Y spacing** fields.
 - **Rectangular Snap** keeps standard snap, as opposed to isometric snap.
 - **Isometric Snap** turns on isometric snap, which is useful for drawing isometric objects.
 - **Polar Snap** sets snap to polar angles.
3. To accept the changes, click **OK**.



Note The **Angle**, **X Base**, and **Y Base** options were removed from this dialog box in AutoCAD 2007, as was the **Rotate** option of the **SNAP** command. Autodesk notes that the snap angle and origin match the current UCS; relocate the UCS to relocate the snap origin. As an alternative, use the **SNAPANGLE** and **SNAPBASE** system variables to change the values from 0 and 0,0, respectively.

Snap

The **SNAP** command has these options for changing the snap spacing:

1. To set the snap increment, use the **SNAP** command's default option. For example, to change the increment to 12 units (or inches), enter 12, as follows:
Command: **snap** (Press ENTER.)
Specify snap spacing or [ON/OFF/Aspect/Style/Type] <A>: **12**
2. To make the vertical (y) snap increment 10 times that of the horizontal (x) direction, use the **Aspect** option:
Command: **snap** (Press ENTER.)
Specify snap spacing or [ON/OFF/Aspect/Style/Type] <A>: **a**
Specify the horizontal spacing <0.5000>: **1**
Specify the vertical spacing <0.5000>: **10**
3. To select the type of snap mode, use the **Style** and **Type** options. **Style** selects between regular and isometric modes, while **Type** selects between rectangular and polar modes. Here is how to enter polar mode:
Command: **snap** (Press ENTER.)
Specify snap spacing or [ON/OFF/Aspect/Style/Type] <A>: **t**
Enter snap type [Polar/Grid] <Grid>: **p**

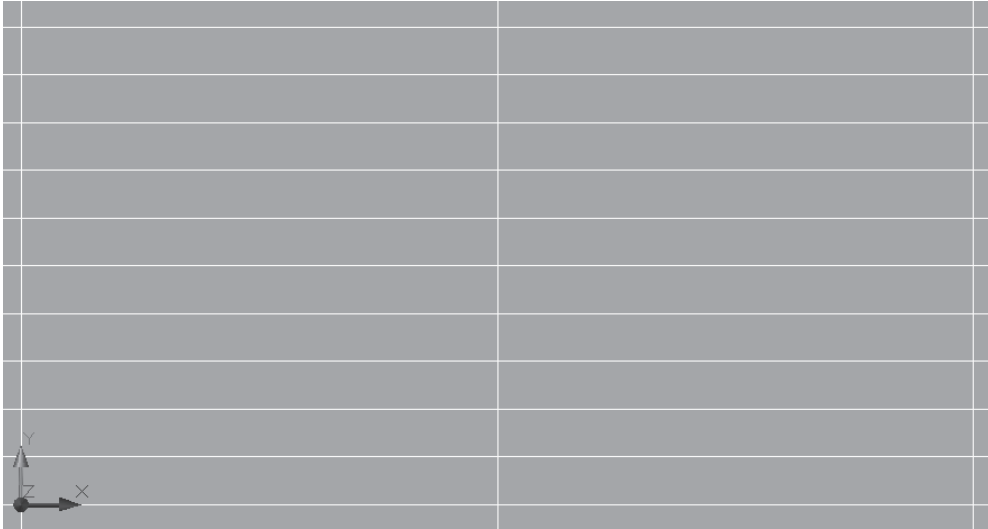
TUTORIAL: DRAWING WITH GRID AND SNAP MODES

Although the grid display is often used together with the snap spacing, the two can be used independently of each other. While it is common to set the snap spacing to the drawing resolution (such as 1/8" or 1 cm), it's better set the grid spacing to a larger value (such as 1' or 1 m), so that the grid doesn't clutter the screen.

In the first part of the tutorial, you turn on and off the grid display, and change its spacing.

1. On AutoCAD's status line, right-click **GRID** (or ) , and then choose **Settings** from the shortcut menu. Notice the Snap and Grid tab of the Drafting Settings dialog box.
2. Select **Grid On** to turn on the grid display, so that a check mark appears in the box. (The grid does not appear until after the dialog box closes.)
3. Set the **Grid X and Y Spacing** to 1.0.
4. Click **OK**. Notice the grid on the screen. If AutoCAD is in 2D wireframe mode, the grid appears as an array of dots; otherwise, it appears as vertical and horizontal lines. (You can switch modes with the **VS** command.)

5. Press **F7**. This key toggles the grid off and on.
Press **F7** again.
6. Return to the Snap and Grid dialog box, and then turn off **Equal X and Y spacing**.
7. Set the **X Grid Spacing** to 10, and then the **Y Grid Spacing** to 1.
Notice that it is not necessary for the vertical and horizontal values to be the same.
8. Click **OK**, and notice the grid spacing has changed, as illustrated below.
(Depending on the mode AutoCAD is in, you may see grid dots instead of lines.)



In the following steps, you draw a few lines with snap mode turned on.

9. Right-click the word **SNAP** (or ) on the status line, and then choose **Settings**.
10. In the Snap and Grid tab, choose **Snap On** to turn on snap mode.
Ensure the **Equal X and Y spacing** option is turned on, and then change the **Snap X** and **Y Spacing** to 0.25.
Click **OK**. You won't notice any difference in the display until you start drawing.
11. Start the **LINE** command, and then pick endpoints on the screen. Notice the crosshair cursor "snapping" to specific points on the screen.

Next, you make the grid spacing equal to the snap spacing.

12. Right-click **GRID** on the status line, and then choose **Settings**.
Set the grid spacing to 0.25. The grid display has the same spacing as the snap.
Click **OK** to exit the dialog box.
13. Use the **LINE** command to draw some lines. Notice that the crosshair cursor now lines up with the grid points.

Now you practice drawing rectangles, one with snap turned off, and another with it on.

14. Use **F9** to turn the snap mode off and on.
15. Draw two rectangles with the **LINE** command — one with snap mode on and one with snap mode off. Notice how the endpoints are easier to line up with snap mode on.



Finally, rotate the snap angle with the **SNAPANG** system variable to see how much easier it is for drawing at angles. (Even though you enter system variables at the ‘Command:’ prompt just like commands, system variables are different from commands: they contain settings and do not execute commands.)

16. At the ‘Command:’ prompt, change the snap angle to 45 degrees, as follows:

Command: **snapang**

Enter new value for SNAPANG <0>: **45**

Notice that the **SNAPANG** system variable affects the snap and crosshair cursor. If you are drawing in 2D wireframe mode, the grid dots are also rotated. Grid lines do not rotate (In 3D mode, they disappear).

17. Draw a rectangle with the rotated snap grid and crosshairs. This is an excellent method of drawing objects that have many lines at the same angle.



ORTHO

When turned on, ortho mode (short for “orthographic”) constrains cursor movement to the horizontal and vertical directions. This mode is very useful, because much drafting takes place at right angles — 0, 90, 180, and 270 degrees — as evidenced by the T-squares and right-triangles used in manual drafting.

Ortho mode can be changed from moving only in the horizontal and vertical directions. Use the **SNAPANG** system variable to change the ortho angle, as detailed earlier. Alternatively, use the polar option, described later.

TUTORIAL: TOGGING ORTHO MODE

1. To turn on ortho mode, use the **ORTHO** command by one of these methods:

- From the status bar, click **ORTHO** or  (turned on when button looks blue).
- On the keyboard, press function key **F8** or **CTRL+L**.
- At the ‘Command:’ prompt, enter the **ortho** command.

Command: **ortho** (Press ENTER.)

2. Use the **ON** option to turn on ortho:

Enter mode [ON/OFF] <ON>: **on**

You don’t notice any difference in the cursor movement until you use drawing and editing commands. Then the cursor moves only vertically or horizontally.

3. To turn off ortho mode, repeat the command with the **OFF** option:

Command: **ortho** (Press ENTER.)

Enter mode [ON/OFF] <ON>: **off**

Unlike the **GRID** and **SNAP** commands, the **ORTHO** command has no additional options. Ortho mode is, however, affected by these snap options: angle, base point, and isometric style.



Notes While you can rotate the ortho angle, drawing still takes place at 90 degree increments in most cases. The exception is when isometric mode is turned on: then the ortho angle is 120 degrees.

To draw at other angles, use polar tracking together with PolarSnap. Ortho mode and polar tracking, however, cannot both be on at the same time: turning on one turns off the other. AutoCAD ignores ortho mode in 3D perspective views.

Ortho has generally been supplanted by polar mode, which is more flexible.

TUTORIAL: DRAWING IN ORTHO MODE

1. If ortho mode is on, click **ORTHO** on the status line to turn it off.
2. Draw a rectangle with the **LINE** command. Notice that is hard to draw lines that are perfectly horizontal and vertical.
3. Press **F8** to turn on ortho mode again.
4. Draw another rectangle with the **LINE** command. After picking the first endpoint, notice how the cursor moves precisely horizontally or vertically until you pick the next point.

POLAR

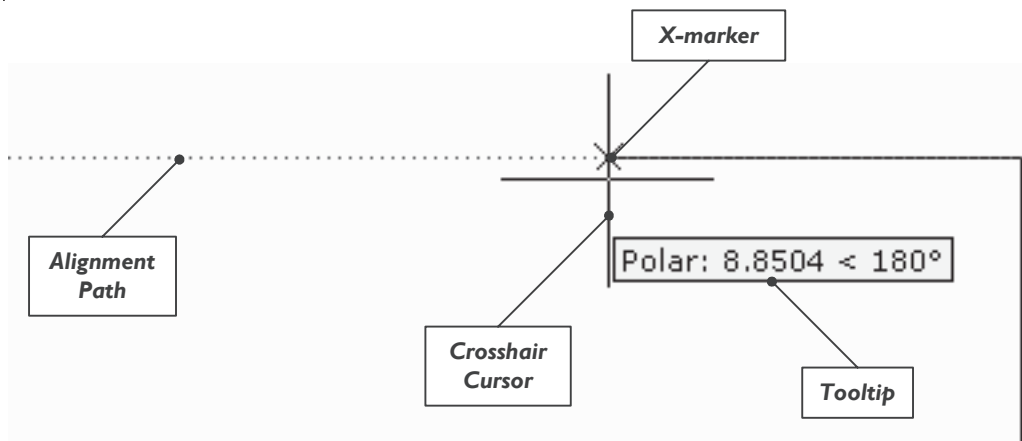
If ortho mode is like manual drafting with a T-square and right-triangle, then polar mode is like drafting with 30-60-90 and 45-45-90 triangles. Polar mode is more flexible than ortho mode, allowing you to draw at 90-degree increments as well as at any other angle, such as 45 degrees or 22.5 degrees to mimic a Vemco drafting machine.

When you turn *on* polar mode, AutoCAD automatically turns *off* ortho mode. That's because ortho limits cursor movement to 90 degrees, while polar is any angle,

TUTORIAL: TOGGLING POLAR MODE

Curiously, there is no “POLAR” command even though AutoCAD has shortcut keystrokes to turn polar mode on and off. (As you later find out, polar mode can be controlled by system variables and dialog boxes.)

1. To turn on polar mode, use one of these methods:
 - From the status bar, click **POLAR** or  (turned on when button looks blue).
 - On the keyboard, press function key **F10** or **CTRL+U**.
2. To initiate polar mode, start the **LINE** command, and then pick a point.
3. At the “Specify next point” prompt, move the cursor around. Notice that every so often a *tooltip* appears, together with an *x-marker* and an *alignment path*, as illustrated below:



Tooltips are small rectangles with explanatory text. In this case, the tooltip displays the polar distance, such as:

Polar: 8.8504 < 180°

This means that the line is 8.8504 units long, at an angle of 180 degrees.

X-marker emphasizes the end of the line and the start of the alignment path. It marks the point where the line would end if you were to click the mouse button.

Alignment path is the thin dotted line that shows you where the line would be drawn, if you were to continue in that direction — almost like a preview.

CONTROLLING POLAR MODE: ADDITIONAL METHODS

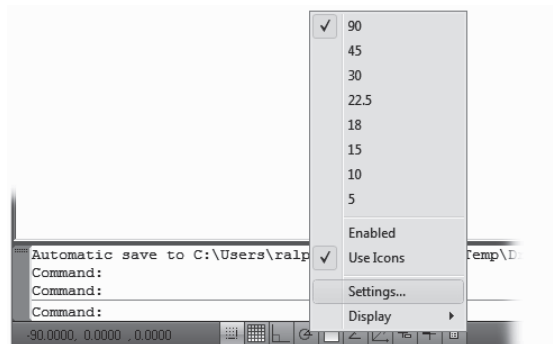
In new drawings, the default polar angle is 90 degrees, the same as ortho mode. You can add angles and specify whether angle measurements are absolute, or relative to the last angle. The changes are made through a dialog box or at the command line using system variables.

- **DSETTINGS** command displays a dialog box for changing snap settings.
- **POLARANG** system variable specifies the increment for polar angles.
- **POLARDIST** system variable specifies the polar snap increment, but only when the **SNAPSTYL** system variable is set to 1.
- **POLARMODE** system variable specifies a number of variables for polar snap.

DSettings

The **DSETTINGS** command displays a dialog box that controls polar mode, because there is no “POLAR” command.

- To access the Drafting Settings dialog box:
 - At the ‘Command:’ prompt, enter the **dsettings** command.
 - Alternatively, right-click **POLAR** on the status line. From the shortcut menu, select **Settings**.



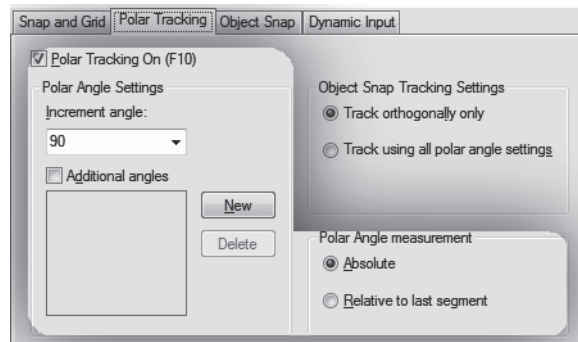
In all cases, AutoCAD displays the Drafting Settings dialog box. Choose the **Polar Tracking** tab.



Note The shortcut menu (shown above) has angle settings that you can access without going to the dialog box:

90, 45, 30, and so on set the polar increment angle to 90, 45, 30 and other degrees.

- In this dialog box, you can make the following changes to the snap settings:

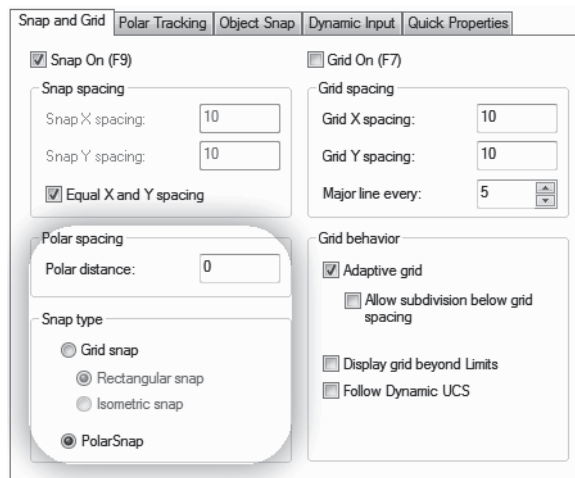


- **Polar Tracking On (F10)** toggles the snap on and off. The “(F10)” reminds you that you can press function key **F10** to turn on and off polar mode.
- **Increment Angle** specifies the polar tracking angle. AutoCAD presets a number of common angles, ranging from 5 to 90 degrees.
- **Additional Angles** allows you to add any angle not provided by the Increment Angle list, such as 7.5. Adding angles is a bit awkward: click the **New** button, enter a value, and then press **ENTER**. Repeat for additional angles.
- **Object Snap Tracking Settings** is an option described later in this chapter.

Polar Angle Measurement switches between absolute and relative angle measurements:

- **Absolute** means that the angle is measured from AutoCAD’s 0-degree, usually the positive x axis.
- **Relative to last segment** means the angle is measured relative to the last-drawn segment. The tooltip changes to read “Relative Polar.”

3. Not all polar settings are listed in this tab of the dialog box. Return to the **Snap and Grid** tab to uncover additional settings.



4. Under Snap Type, select **PolarSnap**. This action causes all the Snap settings to gray out, meaning they are unavailable.
5. Instead, you now specify a distance next to **Polar distance**, such as 2. This sets a snap spacing in the polar direction, i.e., in the direction of the angle specified in the previous tab. (Snap must be turned on.)
6. Click **OK** to exit the dialog box.
7. With the **LINE** command, draw some lines, noticing the action of the cursor: you draw lines at specific angles, and the lines have lengths that are multiples of 2.

In general, you use the Drafting Settings dialog box to control polar mode, but you may prefer the command line at times.

PolarAng

The POLARANG system variable specifies the increments for polar angles.

Command: **polarang**

Enter new value for POLARANG <90>: *(Enter a new value, such as 15.)*

PolarDist

The POLARDIST system variable specifies the polar snap increment (distance), but only when the SNAPTYPE system variable is set to 1. Recall that polar measurements consist of a distance and an angle; this system variable specifies the distance.

Command: **polar****dist**
Enter new value for POLARDIST <0.0>: *(Enter a new value, such as 1.0.)*

PolarMode

The POLARMODE system variable specifies a number of variables for polar snap. It handles eight situations using a single number by means of *bitcodes*. Bitcodes are added up to create a single number.

When all options are turned on, for example, the value of POLARMODE is 15 (which comes from 1 + 2 + 4 + 8). Zero means *no*, *off*, or *not*.

PolarMode	Meaning
Measurement Mode	
0	Absolute mode: Bases polar angle measurements on the current UCS.
1	Relative mode: Measures polar angles from the last-drawn object.
Object Snap Tracking	
0	Tracks using orthogonal angles only.
2	Tracks using polar angle settings.
Additional Polar Tracking Angles	
0	Does not use additional angles.
4	Uses additional angles.
Acquire Object Snap Tracking Points	
0	Acquires points automatically.
8	Acquires points when user presses SHIFT.

Some values used by this system variable are meant for object tracking, covered later in this chapter.

TUTORIAL: DRAWING WITH POLAR MODE

In the previous chapter, you saw that the POLYGON command draws regular polygons but not irregular ones. An example of an irregular polygon is a 45-degree isosceles triangle, which has two angles at 45 degrees and the third at 90 degrees. In this tutorial, you draw such a triangle with two sides 2.5 units long.

1. Before starting to draw, set up polar mode using the **DSETTINGS** command.
In the Drafting Settings dialog box, make these changes:

Polar Tracking tab:

Polar Tracking On

Increment Angle

Polar Angle Measurement

On

45 degrees

Absolute

Snap and Grid tab:

Snap On

Snap Type

Polar Distance

On

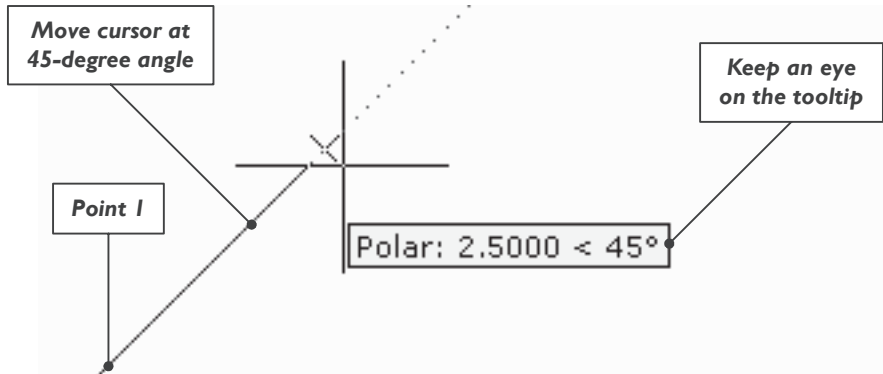
PolarSnap

2.5

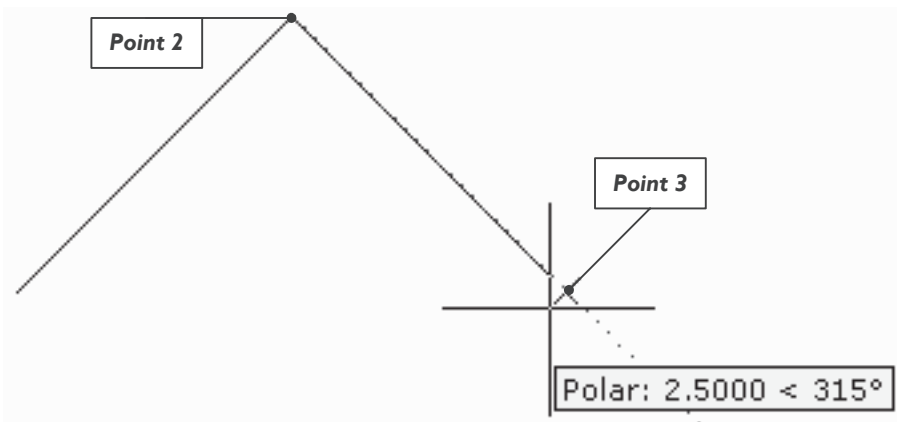
Click **OK** to exit the dialog box.

2. Start the **LINE** command, and then pick a point anywhere in the drawing:
Command: **line**

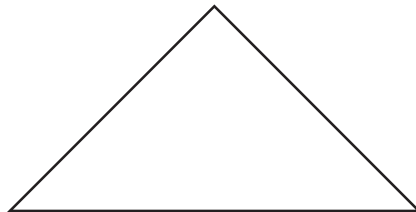
Specify first point: (Pick point 1.)



3. Move the cursor at a 45-degree angle to the upper-right. You may need to move the cursor around until the tooltip appears, reporting **Polar: 2.5000 < 45°**.
Specify next point or [Undo]: (Pick point 2.)
4. Now move the cursor at a 45-degree angle to the lower-right. Click when the tooltip reports **Polar: 2.5000 < 315°**. (The angle of 315 comes from 45 + 270.)
Specify next point or [Undo]: (Pick point 3.)



5. Close the triangle using the **Close** option:
Specify next point or [Close/Undo]: **c**
The isosceles triangle is complete. AutoCAD determines the length of the third side.



OSNAP

Object snap is *the most important tool* for drawing accurately in AutoCAD. You might be able to do without the precise drawing aids described up to this point, but not without object snap. As implied by its name, object snap causes AutoCAD to snap to objects. Object snaps are often called “osnaps” for short.

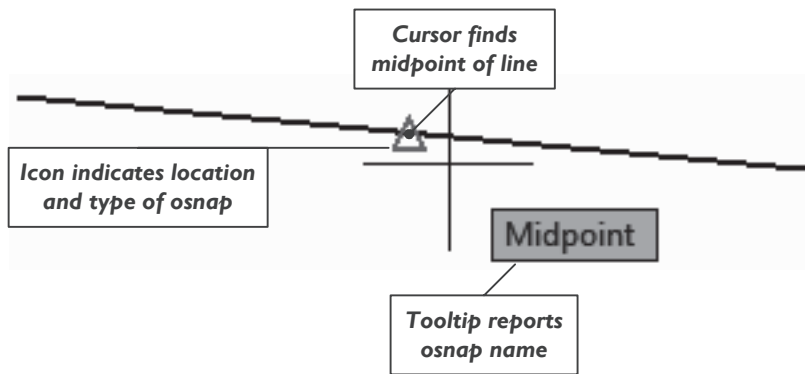
More precisely, AutoCAD’s cursor selects geometric features of objects, such as the ends of arcs, centers of circles, and the intersection of two lines.

In all, there are 13 types of object snaps, as illustrated by the figure at right. When osnap is active, AutoCAD displays temporary icons at the geometric snap point in the drawing. (See figure at right.) In addition, the names of object snaps are listed in shortcut menus — further proof of their importance to accurate drafting.

APERTURE

A powerful aspect of object snap is that you don’t have to be right on the geometric feature — you just have to be close enough (as in horseshoes, hand grenades, and dancing) — and AutoCAD finds a point to which to snap.

The close-enough distance is called the *aperture*, which is a square ten pixels in size around the crosshair cursor. AutoCAD examines the objects that lie within or cross the aperture for likely object snap connections.



- Endpoint
- Midpoint
- Center
- Node
- Quadrant
- Intersection
- Extension
- Insertion
- Perpendicular
- Tangent
- Nearest
- Apparent intersection
- Parallel

You can change the size of the aperture, from as large as 50 pixels to as small as 1 pixel with the APERTURE command; I find that the default of 10 pixels is best.

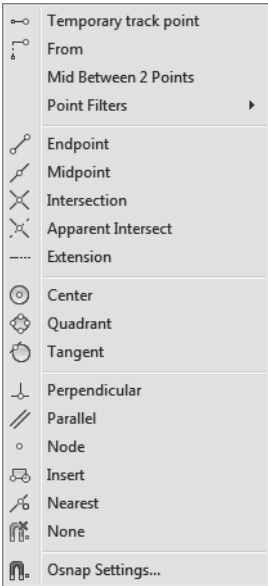
RUNNING AND TEMPORARY OSNAPS

Object snaps are used in two ways during drawing and editing commands: running and temporary. *Running* object snaps are set with the OSNAP and -OSNAP commands; in effect, the object snaps run on and on, until you turn them off.

Temporary object snaps are in effect for the next object selection only. At prompts such as “Select first point:” or “Specify next point”, enter the names of one or more object snap modes. (When entering more than one mode, separate them with commas.) Here a line is drawn from the endpoint of one object to the midpoint or center point of another. “End” and “mid,cen” are abbreviations for the endpoint, midpoint, and center object snaps.

Command: **line**
Specify first point: **end**
of (Pick a point.)
Specify next point or [Undo]: **mid,cen**
of (Pick a point.)
Specify next point or [Undo]: (Press ENTER to end command.)

During commands, you can select object snap modes from a shortcut menu: hold down the CTRL key, and then press the right mouse button (or SHIFT+right-click).



SUMMARY OF ALL OBJECT SNAP MODES

In the review of object snap modes below, notice that the first three letters of each mode are capitalized. These three letters are the abbreviation for each mode. When you enter object snaps at the command line, you need enter only the three letters, not the entire name.

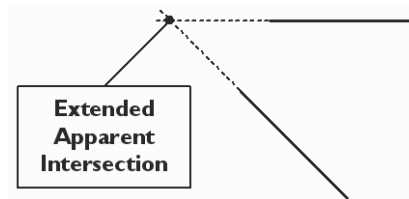
In alphabetical order, AutoCAD's object snap modes are:



APParent intersection

The apparent intersection object snap (called "APPint" for short) works differently, depending on whether you are working with z coordinates.

In 2D drawings, APPint turns on *extended intersection* mode, so that AutoCAD snaps to the intersection of two objects that don't physically intersect. AutoCAD creates an imaginary extension to the two objects, and then determines if an intersection could occur.

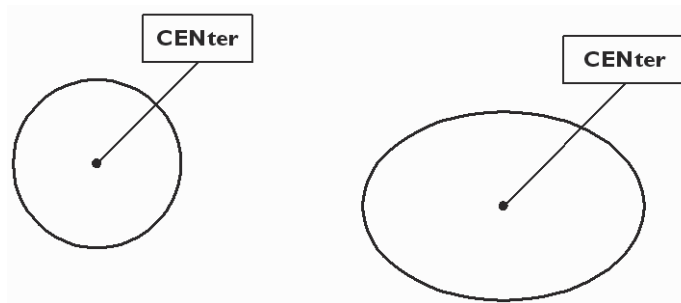


In 3D drawings, APPint snaps to the point where two objects *appear* to intersect from your viewpoint; the objects do not actually need to intersect. The objects can be arcs, circles, ellipses, elliptical arcs, lines, multilines, polylines, rays, splines, and xlines.



CENter

The center object snap snaps to the center point of arcs, circles, ellipses, and elliptical arcs.



ENDpoint

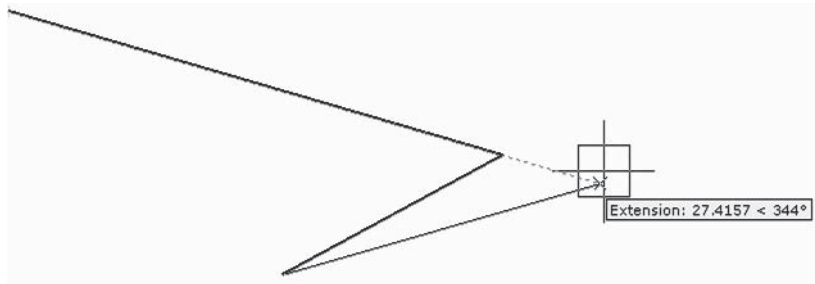
The endpoint object snap snaps to the closest endpoint of arcs, elliptical arcs, lines, multilines, polyline segments, splines, regions, rays, and the closest corner of traces and 2D solids. Use ENDpoint for the corners of 3D faces and solids, because INTersection and EXTended intersection don't work with them.



(History: Early versions of AutoCAD used the END command to exit the program. However, users found that they mistakenly exited AutoCAD when they intended to toggle endpoint object snap by entering 'end.' Autodesk solved the problem by removing the END command, replacing it with QUIT.)

EXTension

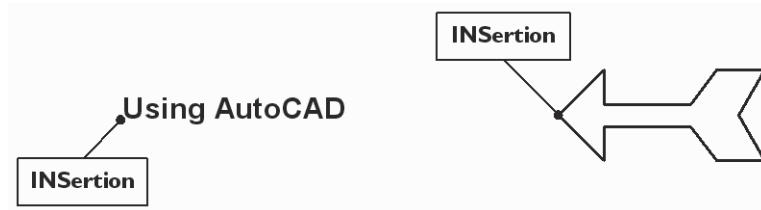
The extension object snap displays an extension line when the cursor passes over the endpoints of objects. This allows you to draw objects that start and end on the extension line.



In the figure above, AutoCAD displays the extension line (shown dotted) from the endpoint of the line. The tooltip reports the distance and angle. (*Extensions* are similar to, but a more limited form of, object snap tracking, discussed later in this chapter.)

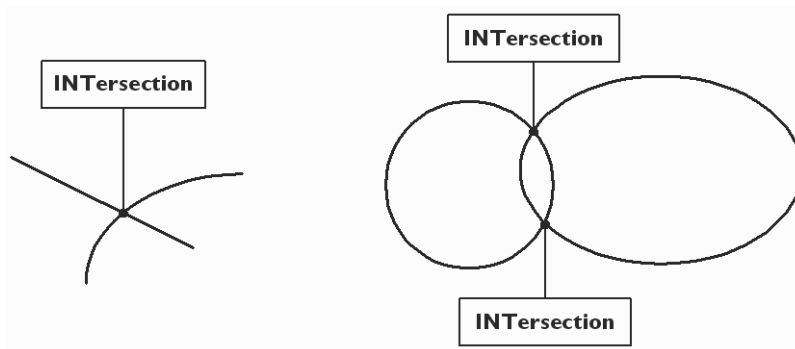
INSertion

The insertion object snap snaps to the insertion point of attributes, blocks, shapes, and text.



INTersection

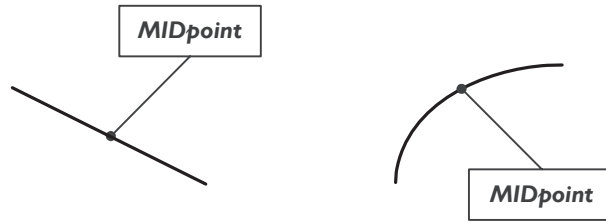
The intersection object snap snaps to the intersection of arcs, circles, ellipses, elliptical arcs, lines, multilines, polylines, rays, regions, splines, and xlines. Sometimes, more than one intersection is possible, such as when two circles intersect.



If extended intersection mode is turned on, AutoCAD can snap to the intersection of two objects that don't physically intersect. An imaginary extension to the two objects is created to determine if an intersection could occur.

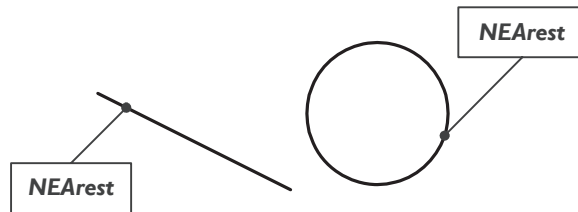
MIDpoint

The midpoint object snap snaps to the midpoint of arcs, ellipses, elliptical arcs, lines, multilines, polyline segments, regions, solids, splines, and xlines.



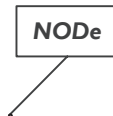
NEArest

The nearest object snap snaps to the nearest point on the nearest arc, circle, ellipse, elliptical arc, line, multiline, point, polyline, ray, spline, or xline.



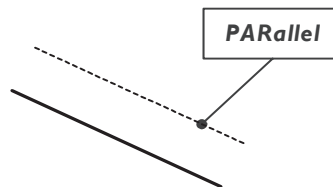
NODe

The node object snap snaps to points, dimension definition points, and dimension text origins.



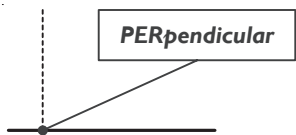
PARallel

The parallel object snap displays an alignment path parallel to a straight line segment, such as a line or polyline.



PERpendicular

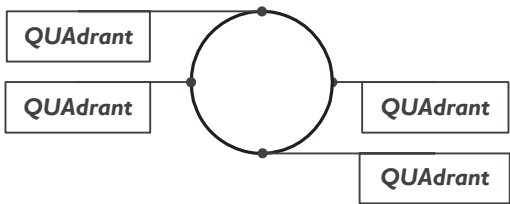
The perpendicular object snap snaps to a point perpendicular to an arc, circle, ellipse, elliptical arc, line, multiline, polyline, ray, region, solid, spline, or xline. Objects can be at any angle.



If necessary, AutoCAD turns on *deferred perpendicular* mode automatically when you need to pick more than one point to determine perpendicularity, such as making a line perpendicular to an arc.

 QUAdrant

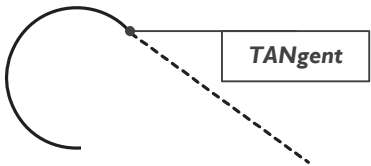
The quadrant object snap snaps to the *quadrant points* of arcs, circles, ellipses, and elliptical arcs.



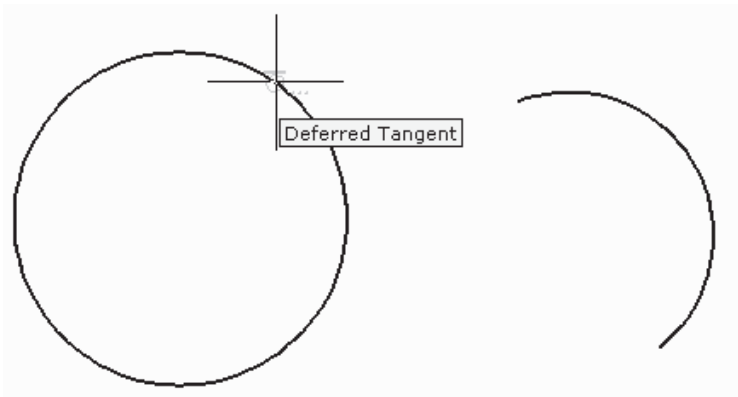
The quadrant points are located at the 0-, 90-, 180-, and 270-degree points of these curves. Thus, circles and ellipses always have four possible quadrant points, while arcs and elliptical arcs can have anywhere from one to four quadrant points.

 TANgent

The tangent object snap snaps to the tangent points of arcs, circles, ellipses, elliptical arcs, and splines.



If necessary, AutoCAD turns on *deferred tangent* mode automatically when you draw more than one tangent, such as a line tangent to two arcs.



ADDITIONAL OBJECT SNAP MODES

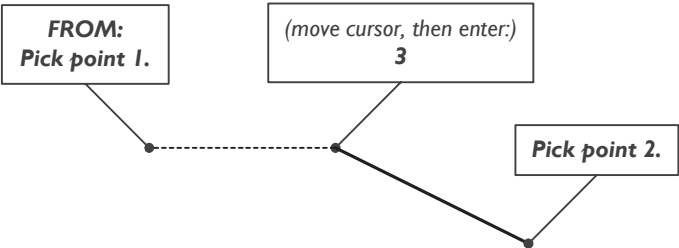
There are several additional object snap modes that don't work with geometry. These find distances, or else turn off snap modes.



From

The from mode allows you to specify a temporary reference or base point, from which to locate the next point.

Command: **line**
Specify first point: **from**
Base point: *(Pick point 1.)*



<Offset>: *(Move the cursor to show the distance, or enter a distance:) 3*
Specify next point or [Undo]: *(Pick point 2.)*
Specify next point or [Undo]: *(Press ENTER to end command.)*



NONE

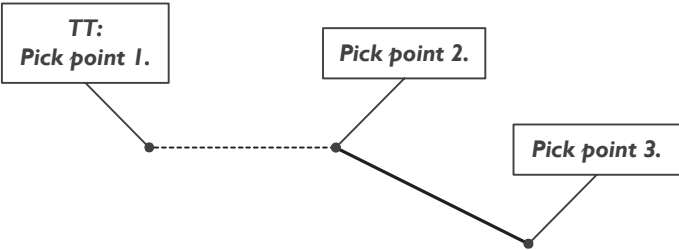
The none mode turns off all object snap modes temporarily for the next pick point.



TT

The temporary tracking mode allows you to specify a *temporary tracking* point. AutoCAD places a + marker at the point, and then, as you move the crosshair cursor, it displays horizontal and vertical alignment paths relative to the + marker.

Command: **line**
Specify first point: **tt**
Specify temporary OTRACK point: *(Pick point 1.)*



Specify first point: *(Pick point 2.)*
Specify next point or [Undo]: *(Pick point 3.)*
Specify next point or [Undo]: *(Press ENTER to end command.)*

QUICK

The **Quick** object snap snaps to the first snap point AutoCAD finds. Quick mode works only when you have two or more other snap modes turned on. For example, when you have intersection and endpoint turned on, AutoCAD returns the first object snap point it finds that is either an intersection or endpoint; this is significant in large, complex drawings. When Quick is off, however, AutoCAD seeks all osnaps, and returns the one closest to the center of the aperture.

OFF

The **Off** option turns off all object snap modes.

TUTORIAL: TOGGLING AND SELECTING OSNAP MODE

Object snap probably has the most complete set of command variations:

1. To toggle ortho mode, use one of these methods:
 - On the status bar, click **OSNAP** or  (turned on when button looks blue).
 - At the keyboard, press function key **F3** or **CTRL+F**.
2. To toggle ortho mode *and* specify one or more object snaps, use one of these methods:
 - During another command, hold down the **CTRL** (or **SHIFT**) key, and then press the right mouse button to display a shortcut menu of object snap modes.
 - Similarly, you can use the Object Snap toolbar during a command: from the toolbar, select a button corresponding to the object snap mode you wish to employ.
 - On the keyboard, enter the **-osnap** command, followed by one or more object snap mode names. (The hyphen prefix forces AutoCAD to display the command at the 'Command:' prompt; without the hyphen, the **OSNAP** command displays the dialog box.)
 - Alternatively, type the aliases **os** or **ddosnap** (the old command name) for the dialog box, and the **-os** alias for the command line.
 - And finally, the **OSMODE** system variable can be used to set object snap modes.

CONTROLLING OSNAP MODES: ADDITIONAL METHODS

In new drawings, AutoCAD remembers the object snap modes set previously. You change the modes through a dialog box, or at the command-line:

- **OSNAP** command displays a dialog box for changing osnap settings.
- **-OSNAP** command changes osnap settings at the command line.
- Several system variables affect object snaps.

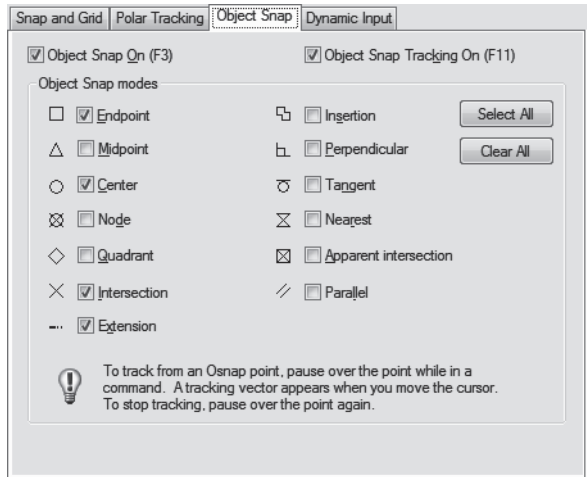


OSnap (DSettings)

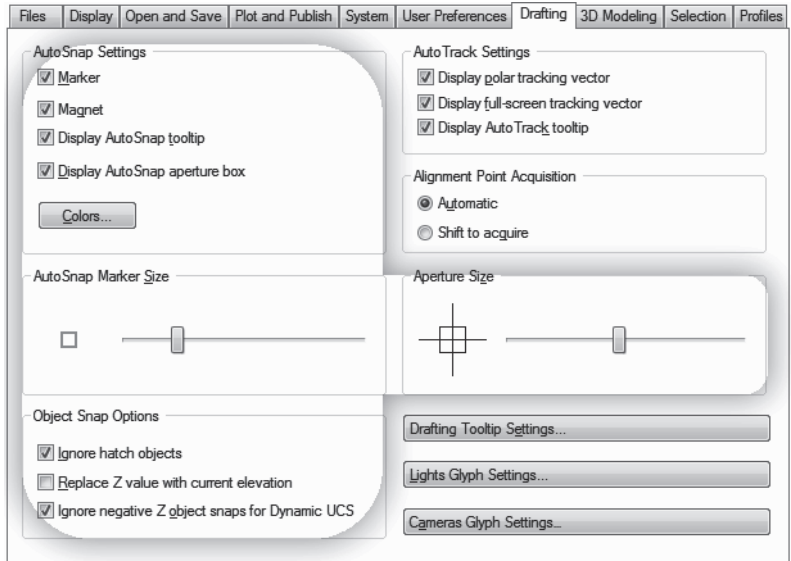
The **OSNAP** command displays a dialog box that allows you to select running object snap modes. (Strictly speaking, the **OSNAP** command is a shortcut for using the **DSETTINGS** command, followed by selecting the Object Snap tab in the dialog box.)

1. To access the Drafting Settings dialog box:
 - At the 'Command:' prompt, enter the **osnap** command.
 - Alternatively, right-click **OSNAP** (or ) on the status bar. From the shortcut menu, select **Settings**.

In all cases, AutoCAD displays the Drafting Settings dialog box. Click the **Object Snap** tab.



2. In this dialog box, you can make the following changes to the snap settings:
 - **Object Snap On (F3)** toggles the snap on and off. The “(F3)” reminds you that you can press **F3** to turn on and off osnaps.
 - **Object Snap Modes** lists most of the modes available. Notice the icon next to each mode; this is the same icon AutoCAD displays in the drawing when it finds a geometric feature matching the selected mode(s).
3. Select one or more object snap modes. You can turn on all osnap modes with the **Select All** button, and turn them all off with the **Clear All** button.
4. Not all object snap options are listed in this dialog box. Click the **Options** button to view more. AutoCAD displays the Drafting tab of the Options dialog box:



Here you select from the many options that affect how object snaps show up in the drawing. For example, if you changed AutoCAD’s background color to white, you may want to change the marker color from the hard-to-see goldenrod to blue or to another higher-visibility color. The options have the following meanings:

Marker toggles the display of the osnap icons.

Magnet toggles whether the cursor automatically moves to the osnap point.

Display AutoSnap Tooltip toggles whether the tooltip appears, which labels the object snap by name.

Display AutoSnap Aperture Box indicates AutoSnap mode is on, through a small

square appearing at the center of the cursor.

Colors selects any color for the marker.

AutoSnap Marker Size makes the marker (icon) larger and smaller.

Ignore Hatch Objects determines whether osnaps snap to the lines that make up hatch patterns.

Replace Z values with current elevation determines whether osnaps use the z coordinate or the value of the **ELEVATION** system variable.

Ignore negative Z object snaps for Dynamic UCS snaps only to positive z-axis locations when dynamic UCS is turned on.

Aperture Size makes the object snap cursor larger and smaller.

5. Click **OK** to exit the dialog box.
6. AutoCAD returns you to the Drafting Settings dialog box. Click **OK** to exit this dialog box.

-OSnap

The **-OSNAP** command displays a prompt at the command line. It lists the current osnap modes, and then prompts you to enter additional modes. You can enter a single osnap mode, such as **Int**, or several modes separated by commas, as shown below:

Command: **-osnap**

Current osnap modes: End,Mid,Cen

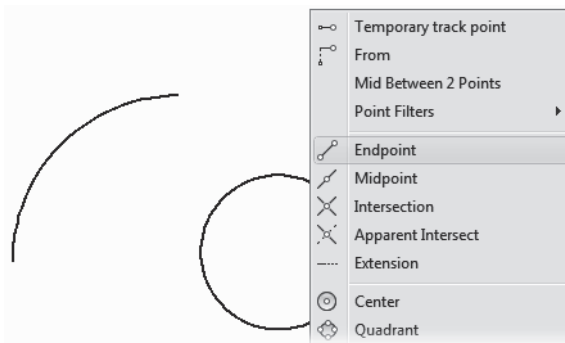
Enter list of object snap modes: **int,qua**

You need only type the first three letters of each mode name. For example, “int” means **INTersection** and “qua” means **QUAdrant**.

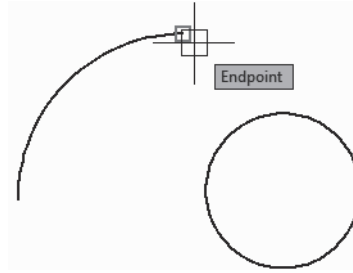
You cannot selectively turn off osnap modes. Instead, you turn off other modes by entering new modes. In the example above, entering **int** and **qua** turns off **end**, **mid**, and **cen**.

TUTORIAL: USING OBJECT SNAP MODES

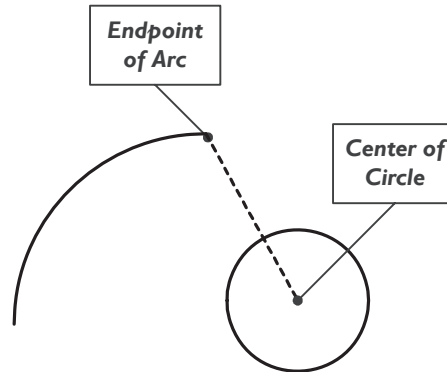
1. Draw a circle and arc of any size.
2. Start the **LINE** command. AutoCAD prompts you:
Specify first point: *(Hold down CTRL key and press right mouse button.)*
3. Do not pick a point. Instead, hold down the **CTRL** (or **SHIFT**) key, and then press the right mouse button to access the shortcut menu.



4. Choose **Endpoint** from the menu. Notice that the aperture square appears at the intersection of the crosshair cursor.
5. Place the cursor over one end of the arc, and click. The line snaps precisely to the end of the arc.



6. Before placing the next point of the line, type **cen**, and press **ENTER**, as follows:
Specify next point or [Undo]: **cen** (Press **ENTER**.)
of (Place the cursor over any part of the circle's circumference, and then click.)
Notice how the line snaps to the precise center of the circle.



7. Continue the line, using the **Center** object snap on the arc. To find the arc's center, you move the cursor to one of these locations:
- Move the cursor onto the arc, and the tracking line appears.
 - Move the cursor roughly where you expect the arc's center to be located.

Either way, the Center icon appears, and when you click, the line snaps to the center point of the arc.



Left: Finding the arc's center at the arc.
Right: Finding the arc's center at its center point.

8. Try other object snap modes on the line, arc, and circle. For example, use the Tangent object snap to construct lines tangent to circles and arcs.

OTRACK

Otrack is short for “object snap tracking”; Autodesk also calls it AutoTrack™, complete with the trademark symbol. Otracking adds alignment paths to the object snap process.

Alignment paths are thin dotted lines that show the osnap's relationship to geometry, as illustrated later.

OSNAP SYSTEM VARIABLES

System variables affect how AutoCAD performs its object snaps.

OSNAPNODELEGACY determines whether NODE osnap snaps to mtext grips:

<i>OSnapNodeLegacy</i>	<i>Comments</i>
0	NODE snaps to mtext grips (default).
1	NODE ignores mtext grips.

OSNAPHATCH determines whether osnap works with hatch objects:

<i>OSnapHatch</i>	<i>Comments</i>
0	Ignores hatch objects (default).
1	Snaps to hatch objects.

DWFOSNAP determines if osnaps works with geometry in DWF files:

<i>DwfOsnap</i>	<i>Comments</i>
0	Ignores DWF entities.
1	Snaps to entities in attached DWF files (default).

DGNOSNAP determines if osnap works with geometry in attached DGN design files:

<i>DgnOsnap</i>	<i>Comments</i>
0	Ignores MicroStation elements.
1	Snaps to elements in imported MicroStation files (default).

PDFOSNAP determines if osnap works with geometry in attached PDF files:

<i>DgnOsnap</i>	<i>Comments</i>
0	Ignores MicroStation elements.
1	Snaps to elements in imported MicroStation files (default).

OSNAPZ determines how osnaps work in 3D:

<i>OSnapZ</i>	<i>Comments</i>
0	Uses the z coordinate of the current point (default).
1	Uses the value stored in the ELEVATION system variable.

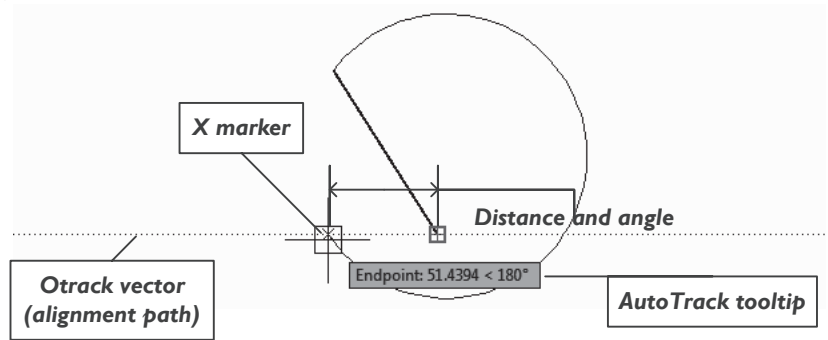
TEMPOVERRIDE determines whether temporary overrides are available:

<i>TempOverride</i>	<i>Comments</i>
0	Does not allow temporary overrides (off).
1	Allows temporary overrides (on).

OSOPTIONS determines how osnaps operate during dynamic UCS mode:

<i>OsOptions</i>	<i>Comments</i>
0	Operates normally during dynamic UCS.
1	Ignores hatch objects.
2	Ignores geometry with negative Z values.

APSTATE reports whether the aperture box cursor is on or off.

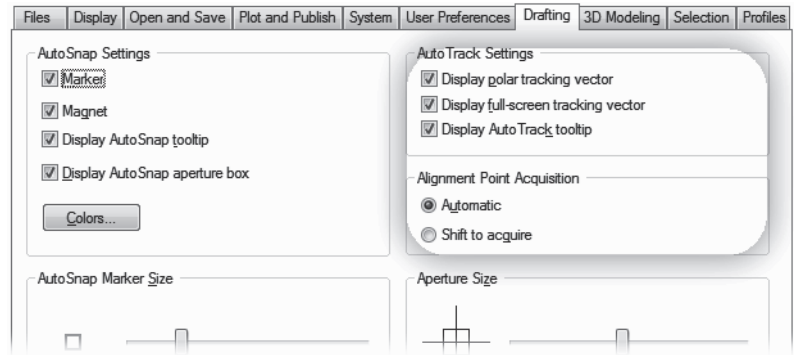


Think of otrack as a “super extension” osnap. At least one object snap mode must be turned on for otracking to work.

When any prompt asks you to ‘Select objects:’, move the cursor around the drawing to see geometric relationships. In the figure above, AutoTrack has placed a marker (the **x**) at the endpoint of the arc and in-line with the endpoint of the line. After a moment, a tooltip appears, reporting the object snap mode and the angle between the geometry.

OTRACK OPTIONS

The Options dialog box lets you control how object tracking looks on your screen:



AutoTrack Settings

Display Polar Tracking Vector displays tracking vectors (dotted lines) at polar angles (angles that divide into 90 degrees) defined by the Polar Tracking tab of the Drafting Settings dialog box.

Display Full-Screen Tracking Vector displays tracking vectors as infinite construction lines.

Display AutoTrack Tooltip toggles the display of AutoTrack and ortho tooltips.

Alignment Point Acquisition

Automatic displays tracking vectors any time the aperture is over an object snap point.

Shift to Acquire displays tracking vectors only when you hold down the **SHIFT** key while the aperture is over an object snap point.

TUTORIAL: TOGGING OTRACK MODE

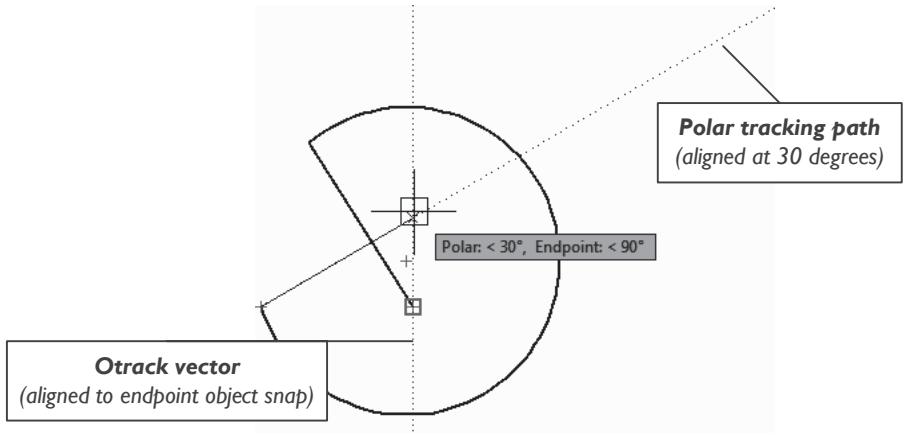
Like polar mode, there is no “OTRACK” command; instead, you use shortcuts, system variables, or dialog boxes to turn it on and off.

To turn on otrack mode, use one of these methods:

- From the status bar, click **OTRACK** or  (turned on when button looks blue).
- On the keyboard, press function key **F11**.

- In the Drafting Settings dialog box’s Object Snap tab, turn on the **Object Snap Tracking On** option.
- Or change the value of the **TRACKPATH** system variable:

TrackPath	Polar Tracking Vector	Object Snap Tracking Vector Is
0	Full screen.	Full screen.
1	Full screen.	Between cursor and alignment point.
2	Not displayed.	Full screen.
3	Not displayed.	Between cursor and alignment point.



You can use otracking together with temporary tracking. At the ‘Select objects:’ prompt, enter **tt** (short for *tracking*) to show alignment paths relative to the temporary tracking point.

The technical editor notes that OTRACK and TT together eliminate the need for the rays, xlines, and other construction lines discussed next.



The **RAY** and **XLINE** commands place *construction lines* in the drawing. Construction lines are useful for creating drawings, helping to reference things like center lines and offset lines. The construction lines, unfortunately, are plotted by AutoCAD, so you should place them on frozen or no-plot layers. **RAY** draws “semi-infinite” lines, which have start points, but no endpoints. **XLINE** draws infinitely-long lines, which have no start or endpoints.

TUTORIAL: DRAWING CONSTRUCTION LINES

1. To draw construction lines, start the **RAY** or **XLINE** command with one of these methods:
 - In the ribbon’s 2D Home panel, choose the **Construction Line** button in the Draw panel.
 - Or, at the ‘Command:’ prompt, enter the **ray** or **xline** commands.
Command: **xline** (Press ENTER.)
 - Alternatively, enter the **xl** alias at the ‘Command:’ prompt.
2. Specify two points that lie along the construction line:
 - Specify a point or [Hor/Ver/Ang/Bisect/Offset]: (Pick point 1.)
 - Specify through point: (Pick point 2.)

TEMPORARY OVERRIDES

Temporary overrides temporarily turn on several object snap and drawing modes. For example, if you are drawing with all osnap modes off, you can hold down **SHIFT+E** to turn on the **END**point osnap mode temporarily.

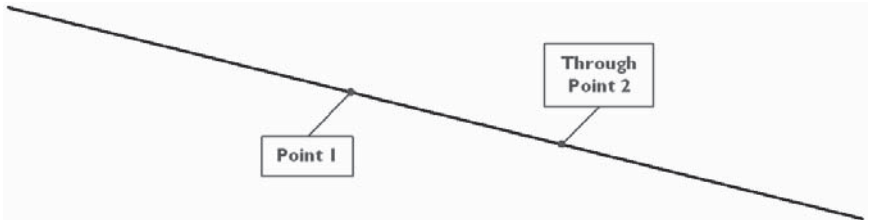
The override does not work in reverse, unfortunately: when **END**point is on, holding down **SHIFT+E** does not turn it off temporarily. The workaround is to press **SHIFT+D** to disable all osnaps temporarily.

Temporary Override	Keyboard Shortcut(s)	
Disable osnap enforcement	SHIFT+A	SHIFT+'
Enable osnap enforcement	SHIFT+S	SHIFT+;
Enable CENter osnap	SHIFT+C	SHIFT+,
Enable ENDpoint osnap	SHIFT+E	SHIFT+P
Enable MIDpoint osnap	SHIFT+V	SHIFT+M
Toggle ortho mode	SHIFT	...
Toggle osnap tracking mode	SHIFT+Q	SHIFT+]
Toggle polar mode	SHIFT+X	SHIFT+.
Disable tracking and all osnap modes	SHIFT+D	SHIFT+L

There are two shortcuts for each, one for left-handed people and the other for right-handed. If none of these temporary overrides works, then the **TEMPOVERRIDE** system variable has been turned off — or, the shortcuts were changed with the **CUI** command.

In addition, the following shortcuts toggle osnap and other modes:

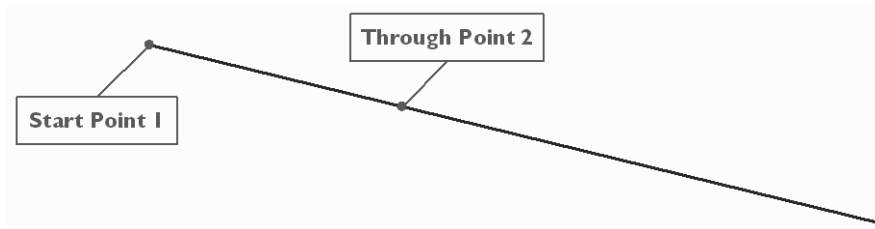
Toggles	Keyboard Shortcut(s)	
Toggle osnap mode	F3	CTRL+F
Toggles ortho mode	F8	CTRL+L
Toggle snap mode	F9	CTRL+B
Toggle polar mode	F10	CTRL+U
Toggle osnap tracking mode	F11	...



- 3. Because the **RAY** and **XLINE** commands automatically repeat, press **ENTER** or **ESC** to exit them:
Specify through point: (Press **ENTER** to exit command.)

The **RAY** command differs from **XLINE** by first asking for the starting point:

- Command: **ray**
- Specify start point: (Pick point 1.)
- Specify through point: (Pick point 2.)
- Specify through point: (Press **ENTER** to exit command.)



DRAWING CONSTRUCTION LINES: ADDITIONAL METHODS

The XLINE command contains these options that provide additional methods for drawing construction lines.

- **Hor** draws horizontal construction lines:

Command: **xline**

Specify a point or [Hor/Ver/Ang/Bisect/Offset]: **h**

Specify through point: *(Pick a point.)*

Specify through point: *(Press ENTER to exit command.)*

- **Ver** draws vertical construction lines:

Command: **xline**

Specify a point or [Hor/Ver/Ang/Bisect/Offset]: **v**

Specify through point: *(Pick a point.)*

Specify through point: *(Press ENTER to exit command.)*

- **Ang** draws construction lines at a specified angle:

Command: **xline**

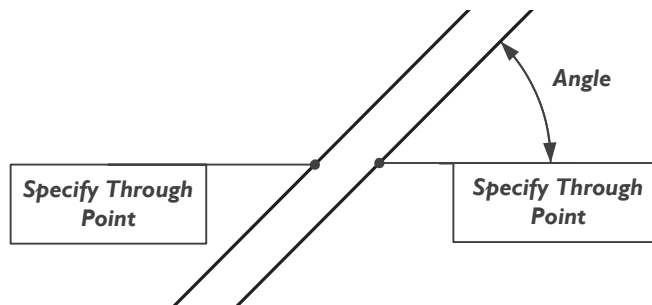
Specify a point or [Hor/Ver/Ang/Bisect/Offset]: **a**

Enter angle of xline (0) or [Reference]: *(Enter an angle, or type r for the reference option.)*

Specify through point: *(Pick a point.)*

Specify through point: *(Pick another point.)*

Specify through point: *(Press ENTER to exit command.)*



- **Bisect** draws construction lines that bisect an angle defined by a vertex and two endpoints:

Command: **xline**

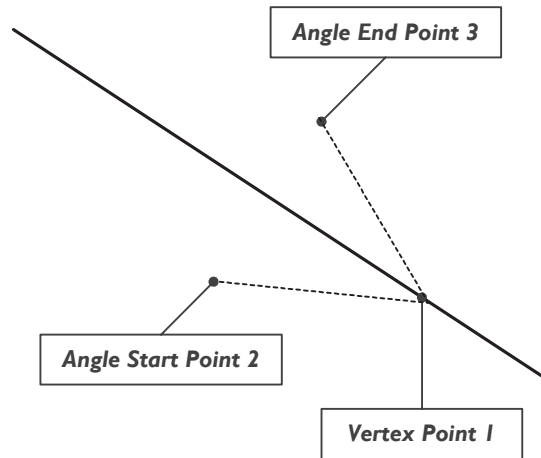
Specify a point or [Hor/Ver/Ang/Bisect/Offset]: **b**

Specify angle vertex point: *(Pick a point.)*

Specify angle start point: *(Pick a point.)*

Specify angle end point: *(Pick a point.)*

Specify angle end point: *(Press ENTER to exit command.)*



- **Offset** draws construction lines parallel to existing lines and construction lines:

Command: **xline**

Specify a point or [Hor/Ver/Ang/Bisect/Offset]: **o**

Specify offset distance or [Through] <Through>: *(Enter a distance, or type t for the through option.)*

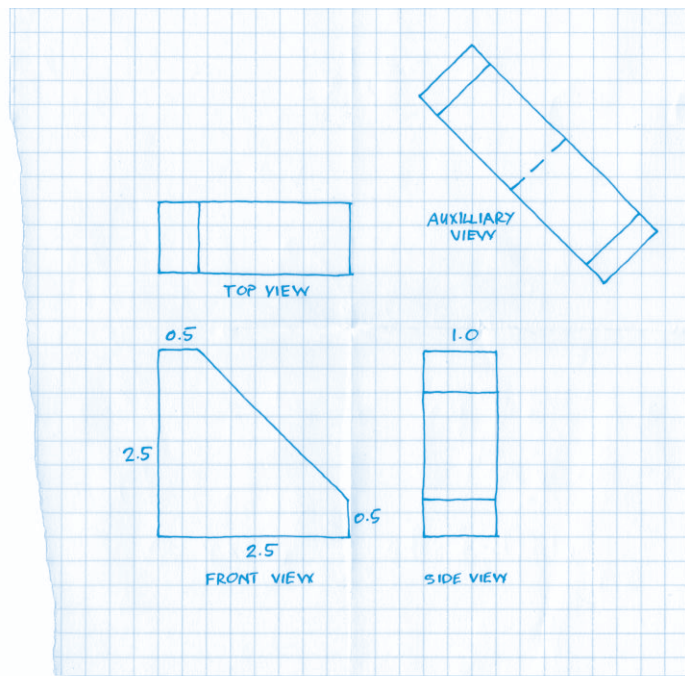
Select a line object: *(Pick a line.)*

Specify side to offset: *(Pick a point.)*

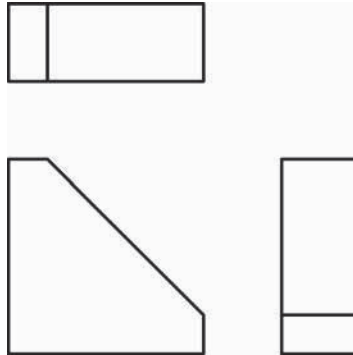
Select a line object: *(Press ENTER to exit command.)*

TUTORIAL: DRAWING WITH CONSTRUCTION LINES

In this tutorial, you draw the true surface of a rectangular inclined face. You use much of what you have learned so far in this chapter: grid, snap, ortho, object snaps, and construction lines.

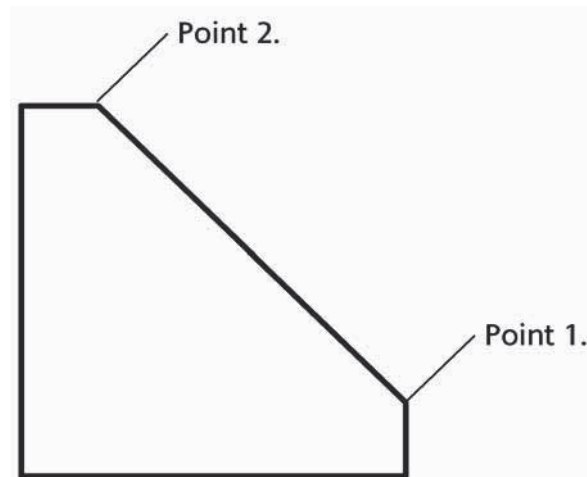


1. Start AutoCAD with a new drawing.
2. Change the following settings:
 Turn on **SNAP**; ensure the snap spacing is 0.5 units.
 Turn on **GRID**.
 Turn on **ORTHO**.
 Turn on **OSNAP**; ensure that object snap modes **Intersection** and **Perpendicular** are turned on.
3. With the **LINE** command, draw the multiview drawing shown below.



Because the top view is a mirror image of the side view, consider using the **MIRROR** command to create it. *Hint:* Turn off **ORTHO** mode, and use a 45-degree mirror line.

4. Use the **SNAP** command's **Rotate** option to rotate the snap and grid to align with the sloping edge of the front view:
 Command: **snap**
 Specify snap spacing or [ON/OFF/Aspect/Rotate/Style/Type] <0.5000>: **r**
 Specify base point <0.0000,0.0000>: (Pick point 1.)



Because you set the object snap modes in Step 2, you should have no difficulty snapping to the ends of the sloped edge.

Specify rotation angle <0>: (Pick point 2.)

Angle adjusted to 315

Notice that the grid changes to align itself with the sloped edge.

5. Turn off **SNAP**.

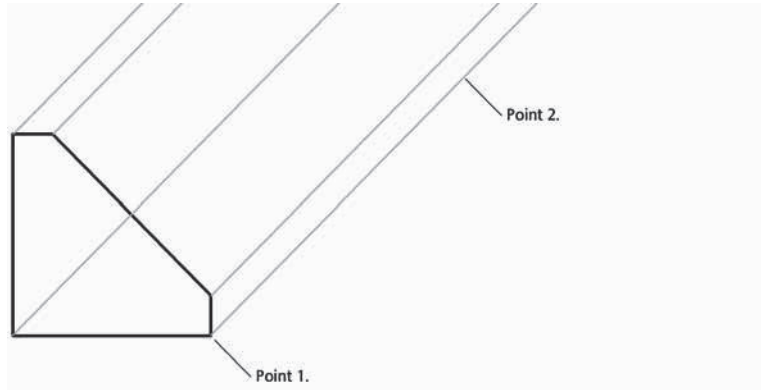
With the **RAY** command, draw the series of parallel construction lines shown below.

Command: **ray**

Specify start point: *(Pick point 1.)*

Specify through point: *(Pick point 2.)*

Specify through point: *(Press ESC.)*



Repeat the command to draw four more construction lines. *Hint: Press **ESC** to end a command that repeats itself; press the spacebar to repeat the command.*

6. Now draw the auxiliary view with the **LINE** command. Remember that the object is 1" wide. *Hint: Use the NEArest object snap to start drawing from the construction line.*

Command: **line**

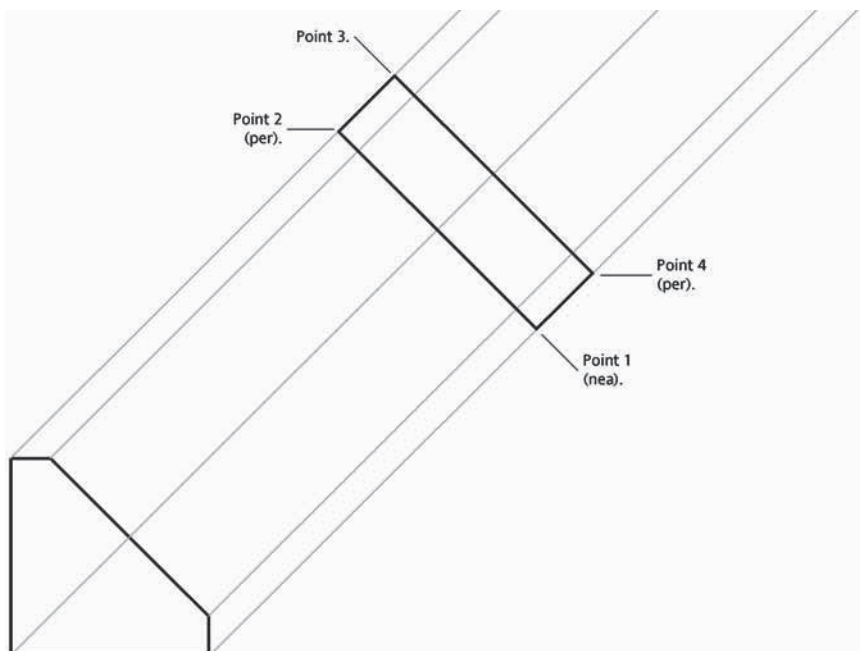
Specify first point: **nea**
to *(Pick point 1.)*

Specify next point or [Undo]: *(Pick point 2.)*

Specify next point or [Undo]: *(Move cursor toward point 3.)* **1**

Specify next point or [Close/Undo]: *(Pick point 4.)*

Specify next point or [Close/Undo]: **c**



7. Draw the two edge lines, as follows:

Command: **line**

Specify first point: *(Pick point 1.)*

Specify next point or [Undo]: *(Pick point 2.)*

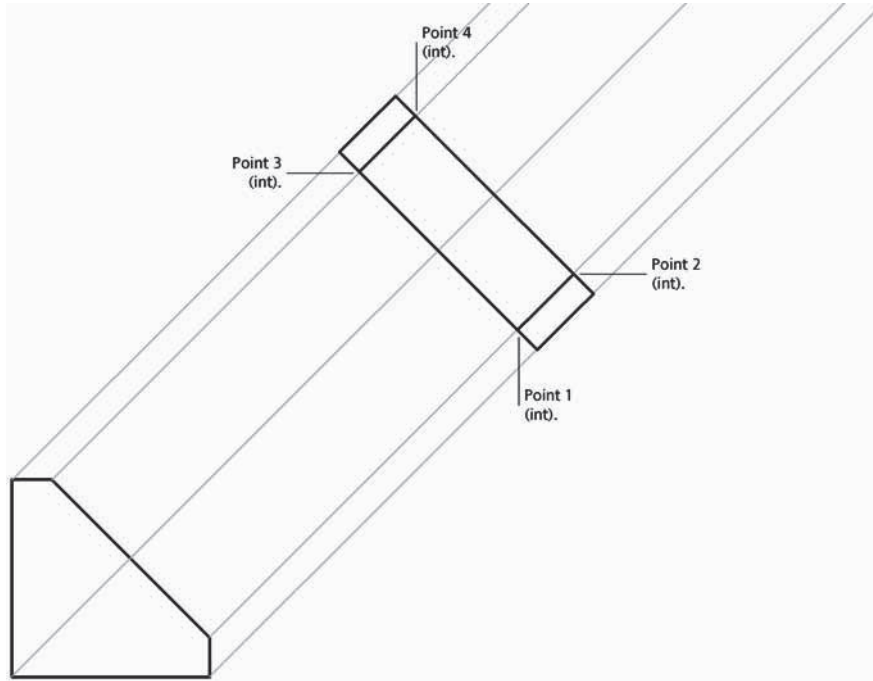
Specify next point or [Undo]: *(Press ESC.)*

Command: *(Press ENTER to repeat the command.)*

LINE Specify first point: *(Pick point 3.)*

Specify next point or [Undo]: *(Pick point 4.)*

Specify next point or [Undo]: *(Press ESC.)*



8. Save the drawing as *auxview5.dwg*.

REDRAW AND REGEN

The **REDRAW** and **REGEN** commands clean up and update the drawing. **REDRAW** cleans up the display by removing blip marks (if any) and blanked out areas. **REGEN** is short for “regeneration,” and updates the display by recalculating all the vectors making up the drawing. During a regeneration, AutoCAD recalculates line endpoints, hatched areas, and so on. A redraw is always faster than a regen.

If you have more than one viewport displayed, **REDRAW** and **REGEN** operate on the current viewport only. Use the **REDRAWALL** and **REGENALL** commands to clean up and update all viewports.

TUTORIAL: USING THE REDRAW COMMAND

- I. To clean up the display, start the **REDRAW** command with one of these methods:

- At the ‘Command:’ prompt, enter the **redraw** command.

Command: **redraw** *(Press ENTER.)*

- Alternatively, enter the **r** alias at the ‘Command:’ prompt.

The screen may flicker as AutoCAD cleans up the drawing.

The commands associated with REDRAW work similarly:

Command	Alias
RedrawAll	ra
Regen	re
RegenAll	rea

TRANSPARENT REDRAWS

A redraw can be executed while another command is active. This is called a *transparent* redraw. To perform this, enter an apostrophe (') before the command. For example, to perform a transparent redraw during the LINE command:

Command: **line**
Specify first point: *(Pick a point.)*
Specify next point or [Undo]: **'redraw**
Resuming LINE command.
Specify next point or [Undo]: *(Pick another point.)*
Specify next point or [Undo]: *(Press ENTER.)*

When the transparent redraw is completed, the previously-active command resumes. REGEN is not a transparent command.

 ZOOM

The ZOOM command enlarges and reduces the view of the drawing. Most drawings are too large and too detailed to work with on the computer’s 17" — or even 21" — screen. CAD operators routinely zoom in to small areas to work on details. When zoomed out, you see more of the drawing; when zoomed in, you see less of the drawing but more detail. Think of zoom as a magnifier and shrinker.

Let’s consider this analogy: imagine that your drawing is the size of the wall in your room. The closer you walk to the wall-size drawing, the more detail you see, but the less you see of the entire drawing. When you move a great distance away from the wall, you see the entire drawing, but not much detail.

The wall does not change size, only your viewing distance. The ZOOM command works in the same way: you enlarge and reduce the drawing size on the screen, but the drawing itself does not change size (or scale).

Because zooming is performed often, AutoCAD has many ways to zoom.

TUTORIAL: USING THE ZOOM COMMAND

- 1. To change the viewing size of the drawing, start the **ZOOM** command with one of these methods:
 - In the ribbon’s 2D Home panel, choose a **Zoom** button in the Utilities panel.
 - On the status bar, click the  Zoom button.
 - Right-click anywhere in the drawing, and from the shortcut menu select **Zoom**.
 - Or, at the ‘Command:’ prompt, enter the **zoom** command.

Command: **zoom** (Press ENTER.)

- Alternatively, enter the **z** alias at the ‘Command:’ prompt.

2. The command displays its many options:
Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/ Previous/Scale/Window/Object] <real time>:

The prompt indicates there are 12 options, and there are more that are hidden: two are **Left** and **Vmax**, and the third is the mouse wheel.

Many of AutoCAD’s commands have a single default option, but Autodesk managed to endow this one with three defaults! These are the three defaults:

(1) Specify corner of window, (2) enter a scale factor (nX or nXP), or (3) <real time>:

Here’s what they mean:

Specify corner of window — click a point on the screen, and AutoCAD executes the **Window** option.

Enter a scale factor — type a number, and AutoCAD executes the **Scale** option.

<real time> — press **ENTER**, and AutoCAD executes real-time zooms.

ZOOM COMMAND:ADDITIONAL METHODS

The ZOOM command contains these options for zooming in and out. The options are shown at the command prompt.

Zoom Option	Meaning
Shows Entire Drawing (zoom out):	
All	Shows the entire drawing, or the limits of the drawing, whichever is larger.
Extents	Shows everything in the drawing.
Vmax	Zooms out to the maximum without invoking a regen (hidden option).
Shows Portion of Drawing (zoom in):	
Center	Zooms about a center point.
Object	Zooms to the extents of the selected objects.
Window	Zooms in to a rectangular area specified by two points.
Left	Zooms relative to a lower left point (hidden option).
Shows All or Portion of Drawing (zoom out or in):	
real time	Zooms in real time with mouse movement.
Previous	Shows the previous view, whether a zoom or pan.
Scale	Zooms by absolute and relative factors in model space and paper space.
Dynamic	Displays a zoom/pan box for interactive zooming (obsolete).

Despite the many options, I find I use only a few frequently — specifically **Extents**, **Window**, and **Previous**. Some CAD users find they don’t need the ZOOM command at all: they use the mouse wheel to zoom in and out of the drawing transparently.

Let’s look at the ZOOM command’s options in alphabetical order.

Zoom All

The **All** option displays the entire drawing on the screen. This typically displays the entire area of the limits and the extents of the drawing, including the area of the drawing outside the limits.

Note this difference between the **Extents** and **All** options: **Extents** displays the extents of the drawing,

while **All** displays either the drawing extents or the drawing limits, depending on which is larger.

Command: **zoom**

Specify corner of window, enter a scale factor (nX or nXP), or
[**All**/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: **a**

Occasionally, **ZOOM All** has to regenerate the drawing twice. If this is necessary, AutoCAD displays the following message on the prompt line:

* * Second regeneration caused by change in drawing extents.

When the limits are changed, the entire drawing area is not shown until the next **ZOOM All** is performed.

Zoom Extents

The **Extents** option displays the drawing at its maximum size on the display screen. This results in the largest possible display, while showing the entire drawing.

Command: **zoom**

Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/**Extents**/Previous/Scale/Window/Object] <real time>: **e**



Note The fastest access to **zoom Extents** is through double-clicking the middle button or mouse wheel.

Zoom Object

The **Object** option displays selected objects at their maximum size on the screen. This option is useful for inspecting one or more objects visually.

Command: **zoom**

Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window/**Object**] <real time>: **o**

Zoom Previous

The **Previous** option returns to the last zoom or pan you viewed. This option is useful if you need to move frequently between two areas. AutoCAD remembers the last ten view changes; simply use the **Z P** aliases several times in a row. Since the view coordinates are stored automatically, you do not need any special procedure to access them.

Command: **zoom**

Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/**Previous**/Scale/Window/Object] <real time>: **p**

Zoom Scale

The **Scale** option enlarges or reduces the entire drawing (original size) by a numerical factor. For example, entering **5** results in a zoom that increases the drawing to five times its normal size — AutoCAD zooms in. The zoom is centered on the screen's center point.

When an x follows the zoom factor, the zoom is computed relative to the current display. For example, entering **5x** makes the drawing five times larger than its current zoom factor.

Only positive values can be used for zoom scales. To zoom out (i.e., display the drawing smaller), use a decimal value smaller than 1. For example, **0.5** results in a view of the drawing that is one-half its normal size. (As of Release 14, **ZOOM All** and **Extents** leave a bit of room around the drawing, effectively doing a zoom 0.95x.)

Command: **zoom**
Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: **s**
Enter a scale factor (nX or nXP): **5**

It is not necessary to enter the “s” for the **Scale** option. Enter a number after starting the command. Then AutoCAD assumes you want the scaled zoom.
The **XP** option scales the model viewport relative to the paper space viewport.

Zoom Window

The **Window** option uses a rectangular window to show the area, which you specify with two screen picks.

Command: **zoom**
Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: **w**
Specify first corner: (Pick point 1.)
Specify other corner: (Pick point 2.)

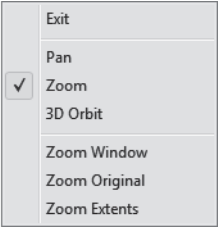
A box is shown around the area to be zoomed.
It is not necessary to enter the “w” for the **Window** option. Simply pick two points after starting the command; AutoCAD assumes you want a windowed zoom.

Zoom Realtime

With today’s computers and fast display drivers providing the horsepower, AutoCAD can zoom in realtime. *Real time* means that the zoom changes continuously as you move the mouse. To enter real-time zoom mode:

Command: **zoom**
Specify corner of window, enter a scale factor (nX or nXP), or
[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: (Press ENTER.)
Press ESC or ENTER to exit, or right-click to display shortcut menu.

Once in real-time zoom mode, the cursor changes to a magnifying glass. To zoom, hold down the left mouse button, and then move the mouse up (toward the monitor) and down (away from the monitor). As you move the mouse up, the drawing becomes larger; as you move the mouse down, it becomes smaller.



While in real-time zoom mode, right-click the mouse to see the shortcut menu:

Option	Meaning
Exit	Exits the ZOOM command.
Pan	Switches to real-time pan mode (discussed later in this chapter).
Zoom	Switches to real-time zoom mode (default).
3D Orbit	Switches to interactive 3D rotation.
Zoom Window	Displays a cursor with small rectangle. Select two points, which AutoCAD displays as the new zoomed view. Remains in real-time zoom.
Zoom Original	Displays the drawing as it was when you began real-time zooming.
Zoom Previous	Displays the extents of the drawing’s objects.

To dismiss the shortcut menu without choosing one of its options, click anywhere outside the menu. When the drawing is the size you want, press ENTER or ESC to exit real-time zoom mode and the ZOOM command.

Rarely Used Zoom Modes

Zoom Center — allows you to specify the center point of the zoom. You then indicate the magnification or height for the new view. If an “x” follows the magnification value, such as 5x, the zoom factor is relative to the current display.

Zoom Dynamic — performs dynamic zooming and panning, and toggles between zoom and pan modes. This option is a predecessor to real-time zoom and pan, and is now rarely used.

Transparent Zoom

A transparent zoom can be executed while another command is active. Enter 'ZOOM at almost any prompt (notice the apostrophe prefix.) There are, however, restrictions:

Fast zoom mode must be turned on, which it is, by default. (This option is set with the VIEWRES command.) You cannot perform a transparent zoom if a regeneration is required, which means you cannot zoom outside the generated area — see **ZOOM Dynamic**.

Transparent zooms cannot be performed when certain commands are in progress. These include the VPOINT, PAN, VIEW, and the ZOOM command itself. Commands on the ribbon use transparent zooms automatically.



PAN

The PAN command moves around the view of the drawing.

Many times, you zoom into an area of the drawing to see more detail. You may want to “move” the screen a short distance, to continue working while remaining at the same zoom magnification. This sideways movement is called “panning.”

A pan is similar to placing your eyes at a certain distance from a paper drawing, and then moving your head about the drawing. This would allow you to see all parts of the drawing at the same distance from your eyes.

TUTORIAL: PANNING THE DRAWING

1. To move the view of the drawing, use the **PAN** command by one of these methods:
 - In the ribbon's 2D Home panel, choose the **Pan** button in the Utilities panel.
 - On the status bar, click the  Pan button.
 - Or, at the 'Command:' prompt, enter the **pan** command.
Command: **pan** (Press ENTER.)
 - Alternatively, enter the **p** alias at the 'Command:' prompt.
2. Hold down the left mouse button; notice the hand cursor.
Move the mouse; notice that the drawing moves with the mouse.
Right-click to see the same shortcut menu as displayed during realtime zoom.
3. To exit pan mode, press **ENTER** or **ESC**.
Press ESC or ENTER to exit, or right-click to display shortcut menu. (Press **esc**.)

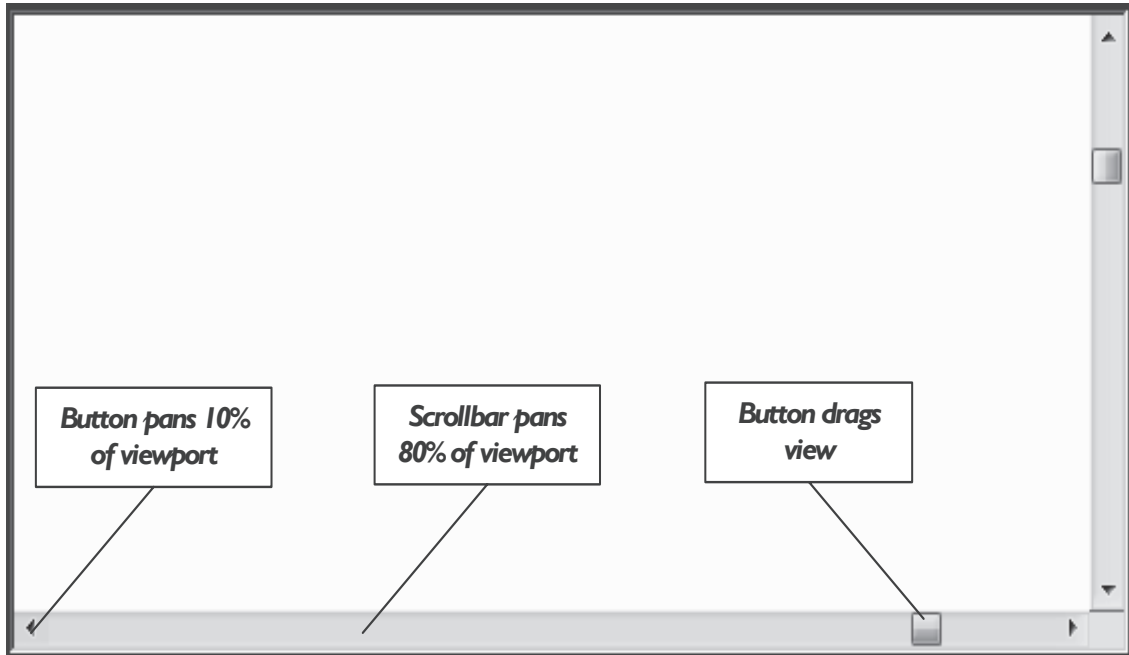
TRANSPARENT PANNING

A pan can be performed transparently while another command is in progress. To do this, enter 'PAN at any prompt, except those expecting you to enter text. The same restrictions apply as for transparent zooms.

Alternatively, you can press-drag the middle mouse button (or mouse wheel).

SCROLL BARS

Like most other Windows applications, AutoCAD has a pair of scroll bars that allow you to pan the drawing horizontally and vertically (though not diagonally). To pan with a scroll bar, position the cursor over a scroll bar and click. AutoCAD pans the drawing transparently.



(If you don't see the scroll bars, you need to turn them on in the Options dialog box. Enter the **OPTIONS** command, choose the **Display** tab, and then turn on the **Display Scroll Bars in Drawing Window** option.)

There are three ways to use the scroll bars:

- Click the arrow at either end of the scroll bar. This pans the drawing in one-tenth increments of the viewport size.
- Click and drag on the scroll bar button. This pans the drawing interactively as you move the button. This is also the way to pan by a very small amount.
- Click anywhere on the scroll bar, except on the arrows and button. AutoCAD pans the drawing by 80 percent of the view.

-PAN

The **-PAN** command is the command-line alternative to panning. It prompts you to pick a pair of points, much like the **LINE** command:

Command: **-pan**

Specify base point or displacement: *(Pick a point.)*

As you move the cursor, AutoCAD draws a dragline.

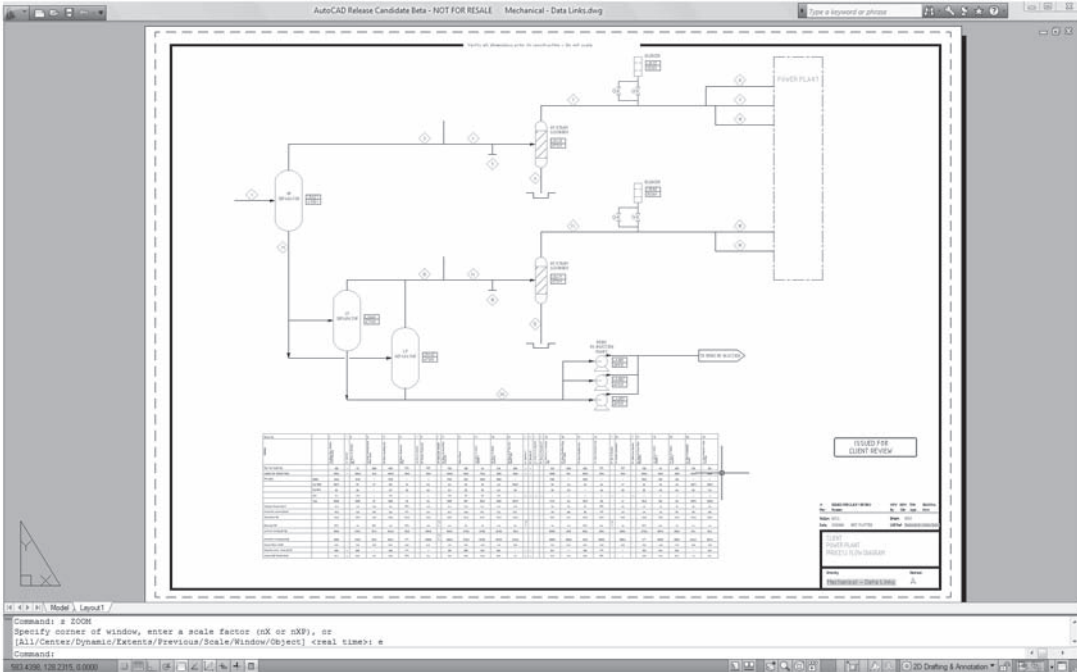
Specify second point: *(Pick another point.)*

After the second pick point, AutoCAD pans by drawing by the distance indicated with the two pick points.



CLEANSCREENON

The **CLEANSCREENON** command minimizes the user interface to maximize the drawing. It turns off the title bar, toolbars, and window edges.



Note You can make the drawing area even larger by turning off the scroll bars and layout tabs. This is done through the **Display** tab of the Options dialog box. In addition, you can drag the Command Prompt away from the bottom of the screen area to make it float, and then make it transparent.

TUTORIAL: TOGGING THE DRAWING AREA

1. To make the drawing area as large as possible, start the **CLEANSCREENON** command:
 - On the keyboard, press **CTRL+0** (zero).

- On the status bar, click the  CleanScreen button.

- At the 'Command:' prompt, enter the **cleanscreenon** command.

Command: **cleanscreenon** (Press ENTER.)

2. To return to the "normal" screen, use the **CLEANSCREENOFF** command.

Command: **cleanscreenoff** (Press ENTER.)

You can press **CTRL+0** (zero) to toggle between maximized and regular views.



VIEW

The **VIEW** command stores and recalls views of the drawing by name. This lets you quickly move about a drawing. Think of it as combining the **ZOOM** and **PAN** commands, without needing to specify zoom ratios and pan directions.

You can save the current display as a view, or else window an area to define the view. Naturally, if you are saving the current view, you need to zoom and pan into that view first. Views can be stored

VIEW TRANSITION

The ZOOM command can employ “view transition.” When zooming in and out, AutoCAD executes an animation that makes objects appear to move closer and further away.

View transitions are useful for seeing where the view “ends up” in the drawing, particularly in 3D; the drawback is that zooms and pans take longer. Real-time zooms still occur in real time; they don’t have the animation.

View transitions are controlled by these system variables:

VTENABLE specifies when view transitions take place (using bit codes):

VtEnable	Meaning
0	View transitions turned off.
1	Turned on for the ZOOM and -PAN commands.
2	Turned on for view rotation.
4	Turned on for scripts.

VTDURATION specifies the duration of view transitions. The default value is 750 milliseconds; range of values is 0 (no transition) to 5000 (very slow transition).

VTFPS specifies speed at which the view transition takes place. The range is 1 to 30fps (frames per second).

GROUPED VIEW-CHANGE UNDO

Undoing a series of view changes, such as repeated pans, can be annoying. The value of 16 in the UNDOCTL system variable groups together all zooms and pans as a single action.

Alternatively, you can turn on the setting in the Options dialog box. From the Tools menu, select **Options**, and then choose the **User Preferences** tab.



relative to the current UCS (user-defined coordinate system), such as when using an architectural plan with an angled wing of the building or when working with 3D drawings.

You take advantage of named views outside of the VIEW command in two ways. When AutoCAD starts, you can specify the name of a view to be displayed when the drawing opens. And, when AutoCAD plots, you can specify a named view with the PLOT command.

Named views can be connected with layer names. That means you can access a view that automatically turns off certain layers for a more specific view. Views can also define visual styles and backgrounds.

The name of views can be changed with the RENAME command, and views can be deleted from drawings with the VIEW command’s “hidden” Delete option.

PREDEFINED VIEWS

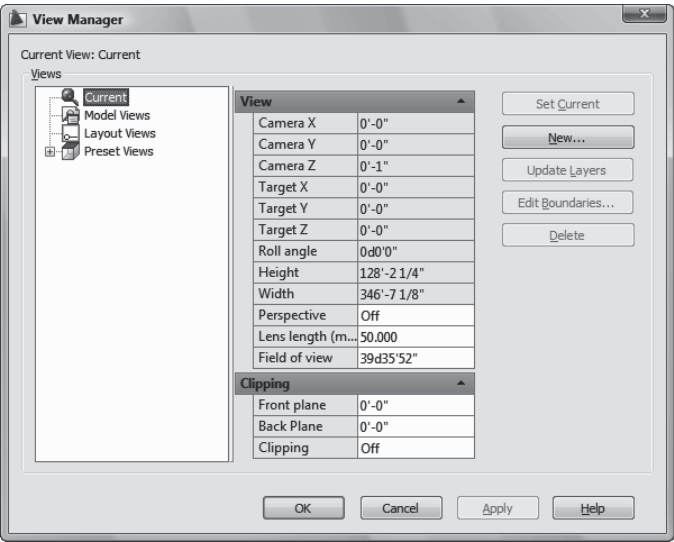
AutoCAD provides several predefined views — six standard orthographic views and four standard isometric views. These are meant for viewing 3D drawings. The orthographic views show the top, bottom, left, right, front, and back of drawings, while the isometric views show the southwest, northwest, northeast, and southeast corners of drawings.

You can access these named views from the View tab’s Views palette on the ribbon.

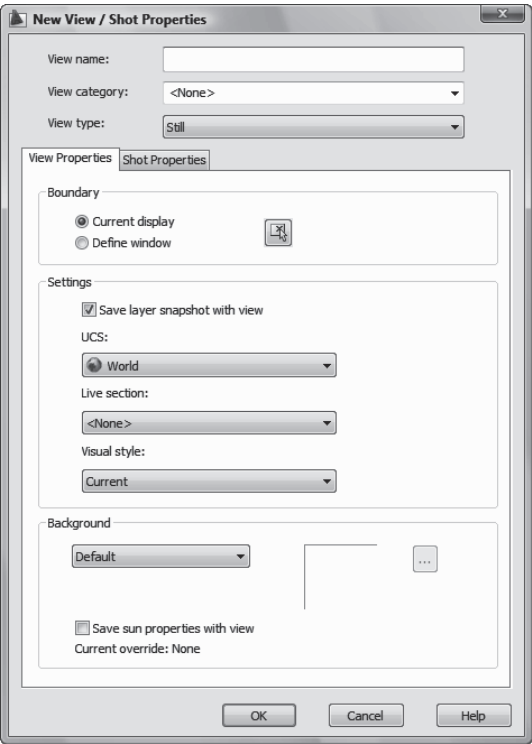
TUTORIAL: STORING AND RECALLING NAMED VIEWS

1. To create named views, start the **VIEW** command with one of these methods:
- In the ribbon's View tab, choose **Named Views**.
 - Or, at the 'Command:' prompt, enter the **view** command.
Command: **view** (Press ENTER.)
 - Alternatively, enter the aliases **v** or **ddview** (the old name for this command) at the 'Command:' prompt.

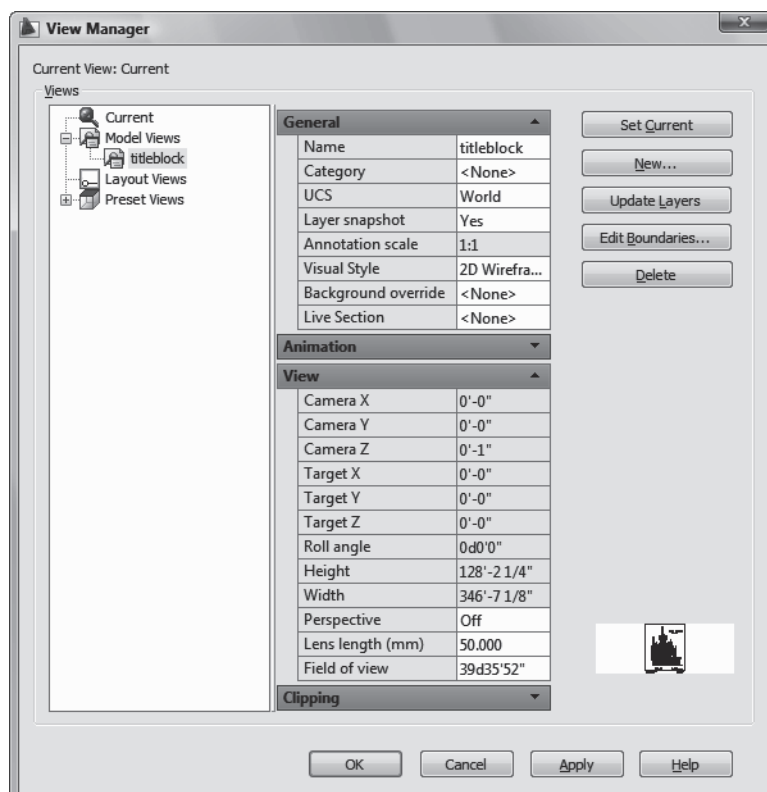
Notice the View dialog box.



2. Click **New** to create a new named view. Notice the NewView dialog box — which you can access directly with the **NEWVIEW** command).



3. In **View Name**, enter a name for the view.
Later, when it comes time to recall the view, you specify it by its name. That's why it makes sense to give views descriptive names, such as "Titleblock."
4. Determine the view's boundary:
 - **Current display** — the view is the current display.
 - **Define window** — the view is smaller than the current display. Click the **Define Window** button. Notice that AutoCAD dismisses the dialog boxes temporarily, and prompts you to pick two points that define the rectangular window.
Specify first corner: *(Pick a point.)*
Specify opposite corner: *(Pick another point.)*
5. Decide whether you want the visibility of layers changed with the view.
When the **Save layer snapshot** option is turned on, AutoCAD remembers which layers were frozen, locked, turned off, and so on. When you later restore the view, the same layers are frozen, etc.
You can ignore the other options — **UCS**, **View Category**, **Background**, and so on — because they are meaningful mainly for working with sheet sets and 3D drawings.
(The Shot Properties tab is for the **NEWSHOT** command's show-motion animation feature, not covered by this book.)
6. Click **OK**.
AutoCAD adds the view name to the list.

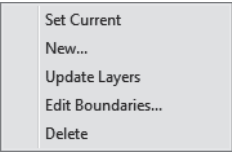


7. To recall the view, select its name (such as "Titleblock"), and then click **Set Current**.
A small triangle points to the current view name.
8. Click **OK**, and notice how AutoCAD fills the screen with a closeup of the drawing's titleblock (or whatever you windowed).

TUTORIAL: RENAMING AND DELETING NAMED VIEWS

You can rename and delete named views through the shortcut menu. Right-click a view name:

- To delete views: select a name, and then click **Delete**.
- To rename views: under the **General** section, click the **Name** field, and then change the name.



The meaning of the items in this shortcut menu is as follows:

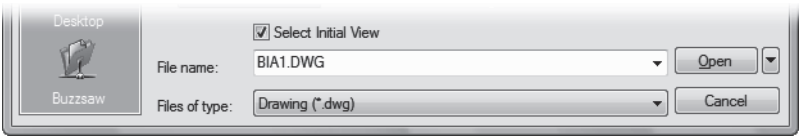
Shortcut Menu	Comments
Set Current	Makes the named view current.
New	Creates new named views.
Update Layers	Changes layer visibility to the current setting.
Edit Boundaries	Shows the windowed view, with non-view areas in gray.
Delete	Removes the named view from the drawing. <i>Warning!</i> AutoCAD removes the view without warning.

TUTORIAL: OPENING DRAWINGS WITH NAMED VIEWS

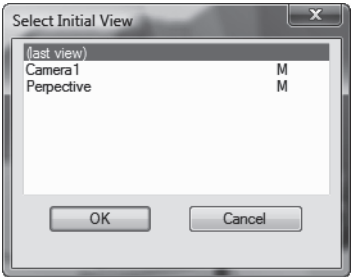
When you use the **OPEN** command, AutoCAD displays the Select File dialog box. One option you may have overlooked is **Select Initial View**, which is normally turned off. When turned on, AutoCAD presents the list of view names, from which you select one.

To open a drawing with an initial view:

1. Enter the **OPEN** command.
2. In the Select File dialog box:
 - a. Select the *bias1.dwg* drawing file from the Companion CD.
 - b. Click **Select Initial View** to turn it on (a check mark shows).



- c. Click **Open**.
3. AutoCAD starts to open the drawing, and then displays the Select Initial View dialog box. Select the “KYOTO” from the list, and then click **OK**.



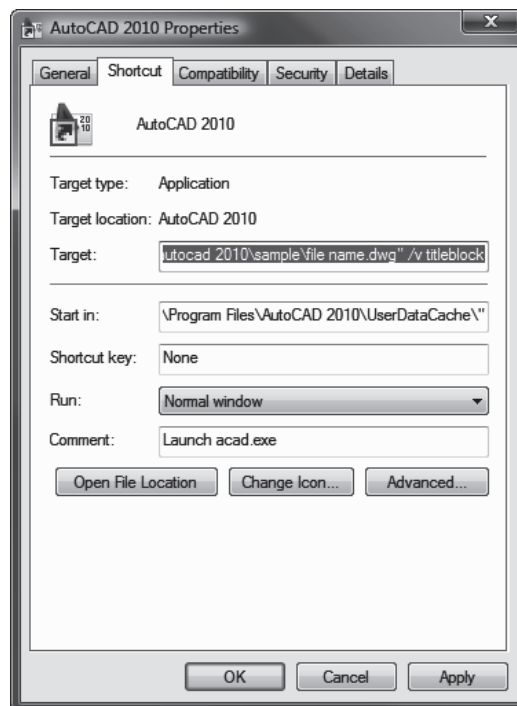
- “M” means the named view was saved in model space.
 - “P” means the view was saved in a layout.
4. AutoCAD finishes opening the drawing, and displays the view.

TUTORIAL: STARTING AUTOCAD WITH NAMED VIEWS

AutoCAD originally ran on another operating system called DOS (short for “disk-based operating system, upon which Windows was also designed.) In those days, it was common to start a program with *command-line switches*, which instructed the program how to start. These switches are still available for AutoCAD, and the `/v` switch specifies the named view to display when AutoCAD opens a drawing. The drawing, naturally, needs to have the named view; otherwise AutoCAD ignores `/v`, and instead shows the last saved view.

To use the `/v` switch, edit AutoCAD’s command line, as follows:

1. On the Windows desktop, right-click the AutoCAD icon.
2. From the shortcut menu, select **Properties**.
3. In the Properties dialog box, select the **Shortcut** tab.
4. In the Target text box, add the path and file name of the drawing. An example is shown in boldface text, but may differ for your computer:
`"C:\AutoCAD 2010\acad.exe" "c:\autocad 2010\sample\file name.dwg" /v titleblock`
5. Click **OK** to close the dialog box, and double-click the AutoCAD icon to see if it loads the drawing with the named view.



Notes A space is required after the switch. Double quotes are needed when there are spaces in the file name.

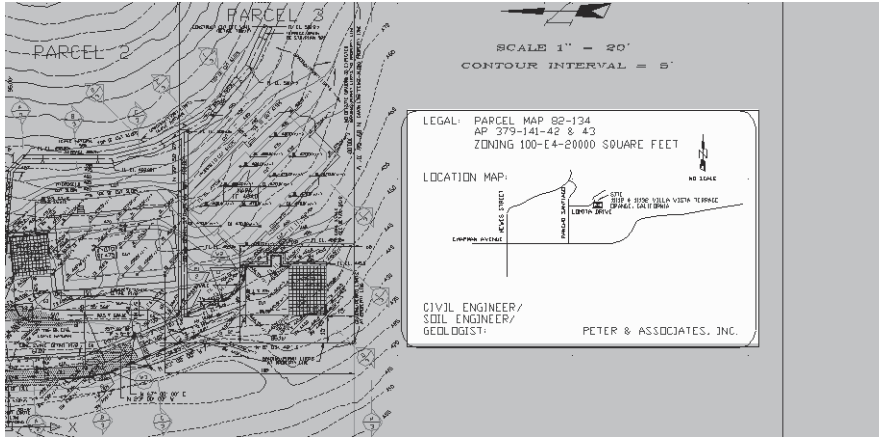
You can have more than one shortcut icon on the Windows desktop. To make copies of icons, hold down the **CTRL** key, and then drag the icon. When you let go, Windows creates the copy, which you can then rename and edit.

TUTORIAL: CHANGING WINDOWED VIEWS

AutoCAD makes it easy to change window views, by showing the current windowing.

1. From the Companion CD, open the *bias1.dwg* file.
2. Enter the **VIEW** command, and then select the “KYOTO” view.
3. Click the **Edit Boundaries** button.

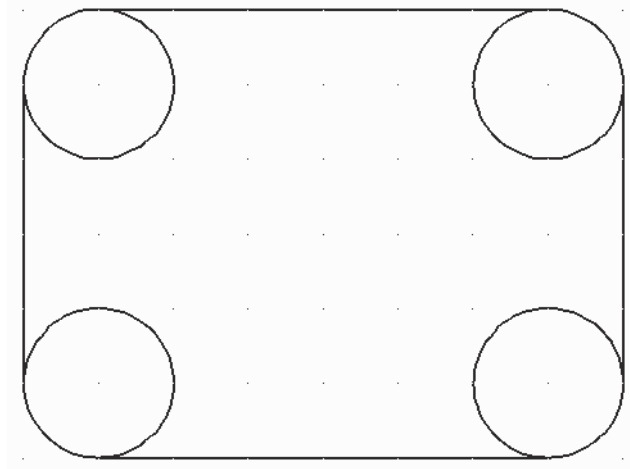
Notice that the dialog box disappears. The area outside the window is shown in gray.



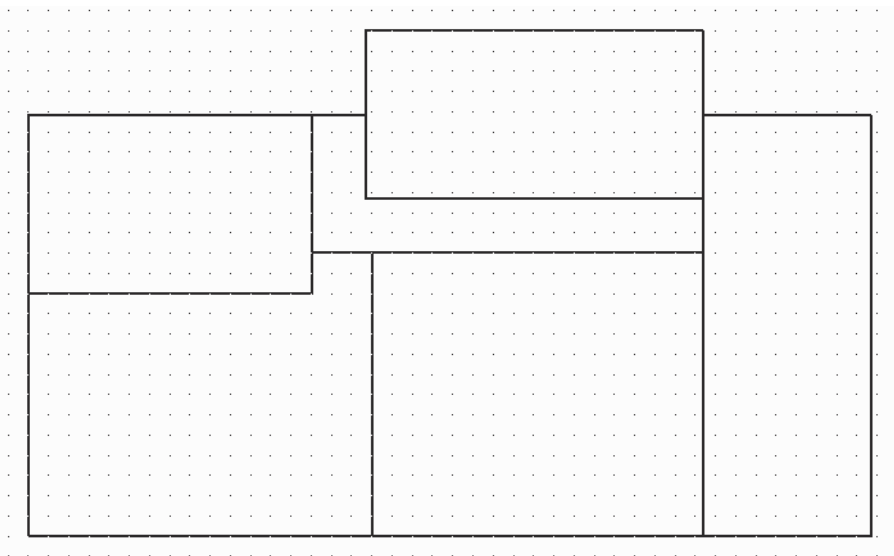
4. At the command prompt, AutoCAD asks you:
Specify first corner: *(Pick a point.)*
Specify opposite corner: *(Pick another point.)*
5. After you pick two points for the new window, AutoCAD shows the area in white (or black, depending on the background color of the drawing area), and asks again:
Specify first corner (or press ENTER to accept): *(Press ENTER.)*
Pick new points, or else press **ENTER** to accept the change.
The dialog box returns.
6. Click **OK** to dismiss the dialog box.

EXERCISES

1. Draw the following baseplate from lines and circles.
Set snap to 0.25 and grid to 1.0. Use the appropriate object snaps to assist your drafting. The dimensions need not be exact, but it may be helpful to know that the circles have a radius of 1.0.



2. Draw the following floor plan using lines.
Set snap and grid to 12.0. Set limits to 0,0 and 600,400. Use the **zoom All** command to see the entire drawing area. Turn on ortho mode. Use the appropriate object snaps to assist your drafting. The dimensions need not be exact, but it may be helpful to know that the longest wall is 500 units long.



3. Draw the following L-shaped bracket using lines and arcs.

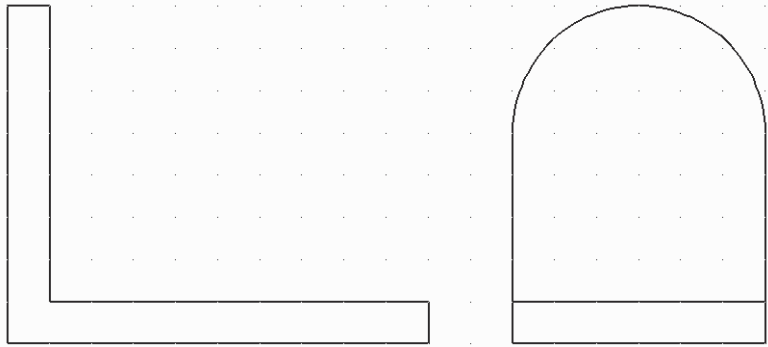
Set the snap to 1.0 and the grid to 12.0.

Set the limits to 0,0 and 216,144.

Use the **zoom All** command to see the entire drawing area.

Turn on ortho mode.

Use the appropriate object snaps to assist your drafting. Remember that arcs are drawn counterclockwise. The dimensions need not be exact, but it may be helpful to know that the longest edge is 10 units long, and that the metal is 1 unit thick.



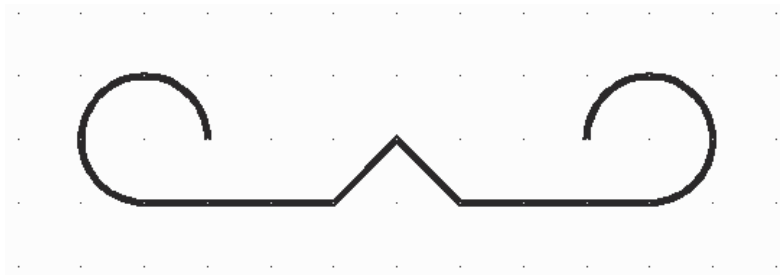
4. Draw an isosceles triangle with lines.

Its angles are 45-45-90, and the two sides are 5 units. Use polar tracking set to 45 degrees and polar snap set to 5 units.

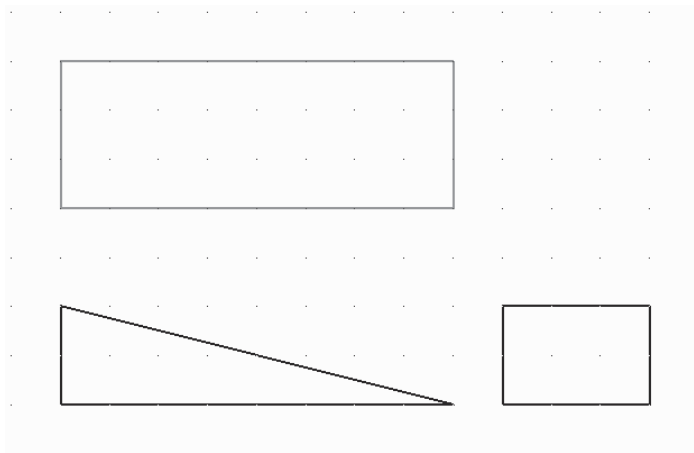
5. Draw the profile of the electrical switch spring shown below.

Use polylines, switching between line- and arc-drawing mode, as necessary.

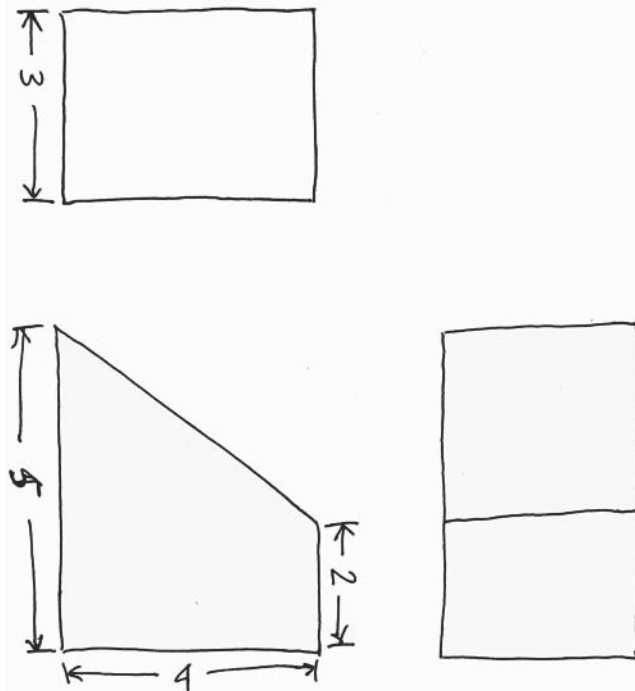
Set the pline width to 0.1 units. Snap and grid are 1.0 units.



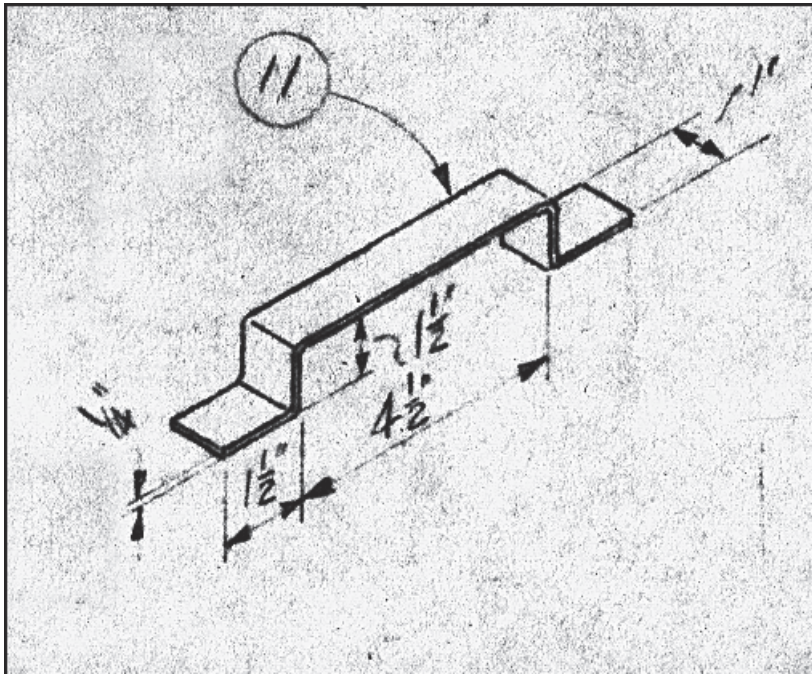
6. Use object snap tracking to assist you in creating the third view (shown in gray) of the industrial strength door wedge. The wedge is 9 units long, 2 units tall, and 3 units wide.



7. Use construction lines to assist you in creating the true view of the object shown below. The units are shown on the sketch.

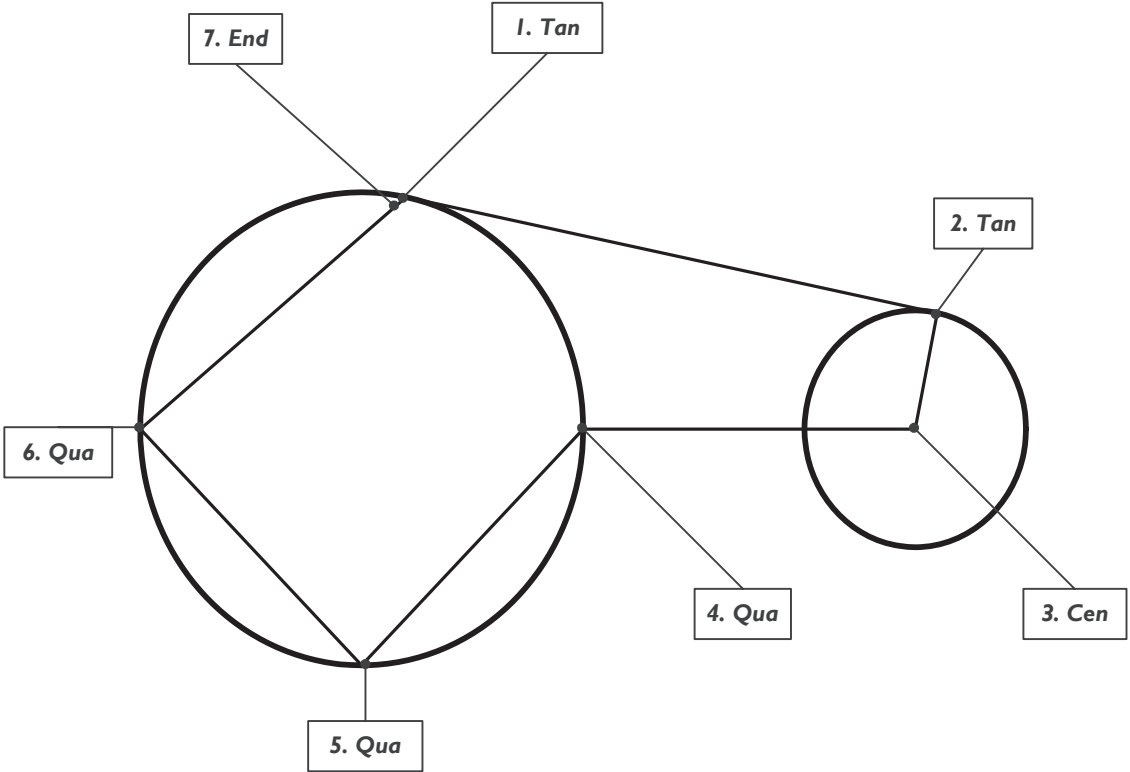


8. Draw the front, side, and top views of the bracket made of 1/4" sheet metal. The units are shown on the sketch.



Drawing courtesy Herbert Grabowski.

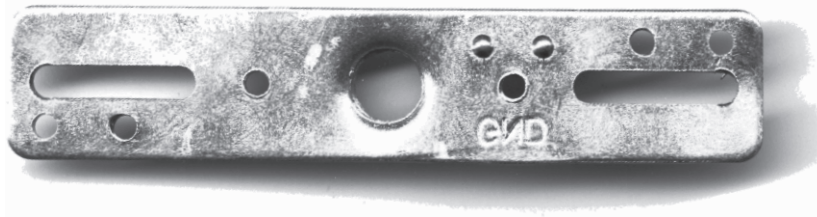
9. Using object snaps, place an arc tangent to the two circles.
Then, draw the remaining lines, using the object snaps shown.
The larger circle has a radius of 4 units, the smaller a radius of 2 units.



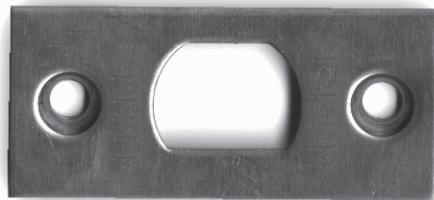
10. Draw the 3.5" wide by 3.7" tall diskette.
The figure below illustrates the diskette full size, 1:1; take your measurements directly from the photograph.



11. Draw the 3.9" wide by 1.75" tall cross brace for ceiling lamps.
Take your measurements directly from the photograph, which is shown below at full-size, 1:1 scale.



12. Draw the 2.25" wide by 1.0" tall latch face for a security deadbolt lock.
Take your measurements directly from the full-size photograph, below.



CHAPTER REVIEW

1. What command allows you to enlarge and reduce the visual size of the drawing?
2. What is the purpose of the **PAN** command?
3. What is the **Realtime** option under the **ZOOM** command used for?
4. When a real-time zoom is in progress, can you override it?
If so, how?
5. How do the **PAN** and **ZOOM** commands differ?
6. What does *toggle* mean?
7. What is a *transparent* command?
How is this option invoked?
8. How do the **REDRAW** and **REGEN** command differ?
9. List four ways by which the grid can be toggled:
 - a.
 - b.
 - c.
 - d.
10. What is *temporary tracking* used for?
11. Name two ways to create *construction lines*:
 - a.
 - b.
12. What is the *aperture*?
13. What object snap mode would you choose to snap to a point object?
14. Can you snap to the midpoint of arcs?
15. When does AutoSnap come into effect?
16. Can different x and y spacings be given to the snap?
Can it coincide with the grid spacing?
17. What advantages do rays and xlines have over grids?
18. What modes provide great accuracy for creating true horizontal and vertical lines?
19. What purpose does **APPint** mode serve?
20. Which command controls the extent of the grid?
21. Although the grid is not a part of the drawing, can it be plotted?
22. Where does a ray start?
End?
23. Name the function of the Aerial View.
24. In addition to the snap angle, the **Angle** option of the **SNAP** command also affects:
 - a.
 - b.
 - c.
 - d.
25. Which mode does **F8** toggle?
26. What is the difference between *ortho* and *polar* modes?
27. What are *tooltips*?
28. When would you use the < symbol in AutoCAD?
29. What is an *alignment path*?
30. Describe the function of *PolarSnap*.

31. Provide the meaning of the following abbreviations:
 - osnap
 - otrack
 - appint
 - tt
32. Define the following object snap abbreviations:
 - int
 - cen
 - qua
 - end
33. Which part of the object geometry do the following object snaps snap to?
 - ins
 - nod
 - per
 - tan
34. When would you use the **From** object snap?
35. Describe how the EXTension object snap operates.
36. Can AutoCAD snap to the intersection of two circles?
If so, in how many places could the object snap take place?
37. Why does AutoCAD sometimes defer snapping to a tangent?
38. Which command is toggled by the following function keys?
 - F3
 - F9
 - F10
 - F11
39. What is the purpose of AutoSnap's magnet?
40. How do the **REDRAW** and **REDRAWALL** commands differ?
41. When might you turn off **REGENAUTO**?
42. Name the command that matches the alias:
 - z
 - sn
 - xl
 - ds
43. Which command maximizes the drawing area?
44. What is the function of the **DSVIEWER** command?
45. Name four commands that take advantage of *named views*.
 - a
 - b
 - c
 - d
46. How do you delete named views?
47. When do you need to increase the setting of the **VIEWRES** command?
48. Describe *view transitions*.
49. How would you turn off view transitions?
50. What keystrokes would you press to turn on MIDpoint object snap temporarily?

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CHAPTER 6

Drawing with Efficiency

In addition to precision, another advantage of CAD over hand drafting is *efficiency*. “You should never have to draw anything twice,” said John Walker, one of the founders of Autodesk, Inc. By drawing efficiently, you complete projects in less time — or more projects in the same time. In this chapter, you learn to use some of AutoCAD’s commands for creating and reusing content, such as symbols (blocks) and hatch patterns:

BLOCK creates reusable components called “blocks.”

INSERT places blocks in drawings.

BASE changes the base point of drawings.

WBLOCK exports blocks and drawings as .dwg drawing files.

BEDIT edits blocks.

HATCH places hatch patterns and solid fills; **GRADIENT** places gradient fills.

HATCHEDIT edits associative hatch patterns and fills.

BOUNDARY creates single regions from disparate areas.

SELECT selects objects by their location; **QSELECT** selects objects by their properties.

DDSELECT controls selection options.

FILTER selects objects based on their properties and location.

DRAWORDER and **TEXTTOFRONT** control the order in which objects are displayed.

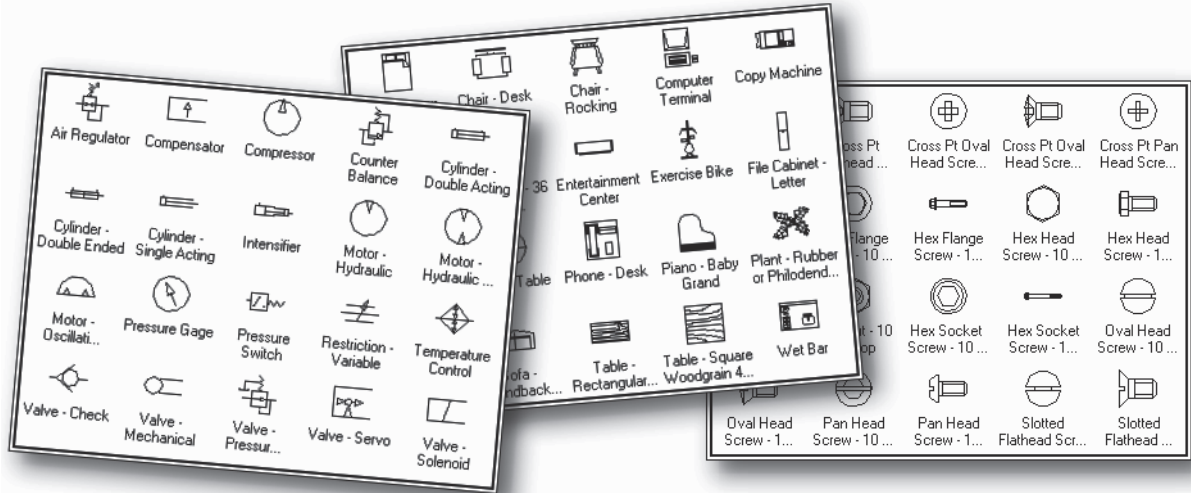
NEW TO AUTOCAD 2010 IN THIS CHAPTER

- **BEDIT** changes its ribbon interface.
- **HATCH** now highlights boundary gaps with red circles.
- The boundaries of non-associative hatch patterns can now be edited.

BLOCK AND INSERT

The **BLOCK** command creates symbols that AutoCAD calls “blocks.” Its complement is the **INSERT** command, which places blocks in drawings. Using blocks in drawings gives you several distinct advantages.

Entire libraries of blocks can be used over and over again for repetitive details. By using pre-drawn blocks, you reduce the amount of drafting required to complete drawings. AutoCAD includes more than a dozen libraries of hundreds of blocks in the `\DesignCenter` and `\Dynamic Blocks` folders, found under `\AutoCAD 2010\Sample`. (See Appendix F for the complete listing.)



Blocks and nested blocks are excellent for building drawings from “pieces.” (A *nested block* is a block placed within another block.) Several blocks require less space than several copies of the same objects, because AutoCAD stores only the information for the original block *definition*.



Note It is more efficient to use blocks than to copy the same objects over and over. The **COPY** command makes a complete copy each time you use it; in contrast, AutoCAD creates a single definition of a block, and then points to it when you make “copies” with the **INSERT** command. This reduces the drawing’s file size and improves the display time.

While standard blocks are static, *dynamic blocks* can be modified interactively according to the designer’s intentions. Dynamic blocks are easily stretched, rotated, mirrored, and aligned to other objects — if the blocks are defined that way. In addition, *lookup tables* allow one block to display itself in multiple forms.

Blocks can be used with *attributes*, which are text records that can be visible or invisible. (Attributes can only be attached to blocks.) The data from attributes can be exported to tables in drawings, and to spreadsheet programs for further analysis. This is useful in facilities management, for example, where multiple occurrences of desks, chairs, and computers are found. Each is stored as a block, and has attributes associated with it, such as the person’s name and telephone number.

AutoCAD stores blocks in the drawing in which they were made. You can share them with other drawings by using DesignCenter, or by exporting blocks as `.dwg` drawing files to disk with the **WBLOCK** command; AutoCAD can import any `.dwg` drawing file as a block.

Editing the Content of Blocks

Because a block is a group of objects combined into a single object, you move, scale, copy, and erase a block as though it were a single object. Indeed, blocks are so “tight” that it is difficult to edit their individual members. The three ways to edit the objects in blocks are as follows, in historical order:

- **EXPLODE** command breaks blocks into their individual parts; when you are done editing the parts, you can use the **BLOCK** command to recombine them.
- **REFEDIT** command “checks out” blocks, and then the **REFSET** command edits them by adding and removing objects; when done, the **REFCLOSE** command adds the changes to the block definitions.
- **BEDIT** command opens blocks in the Block Editor, where the objects can be edited; when done, the **BSAVE** command saves the changes to the original blocks. The **BSAVEAS** command saves the changes to new blocks, preserving the old ones.

TUTORIAL: CREATING BLOCKS

There are three basic pieces of information AutoCAD needs before it creates blocks: (1) a name for the block, so that you can later identify it; (2) x, y, z coordinates for the insertion point, so that AutoCAD knows where to place the block; and (3) the objects that make up the block.

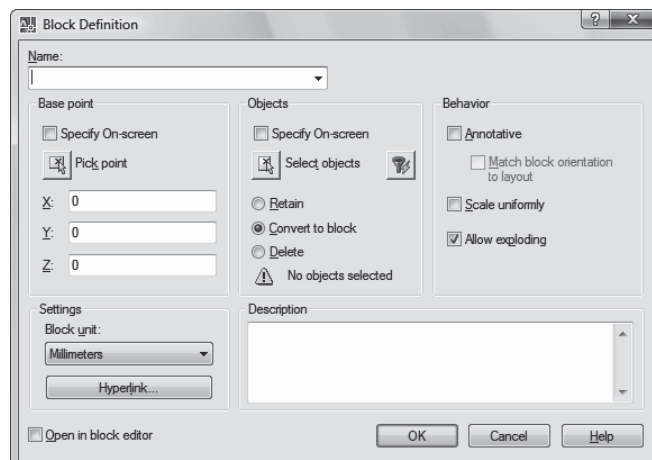
With that in mind, let’s create blocks with the **BLOCK** command. (Dynamic blocks are created with the **BEDIT** command.)

- To create blocks, start the **BLOCK** command with one of these methods:
 - In the ribbon’s 2D Home tab, choose the **Create** button from the Block panel.
 - Or, at the ‘Command:’ prompt, enter the **block** command:

Command: **block** (Press ENTER.)

- Alternatively, enter the aliases **b**, **bmod**, or **bmake** (one of the command’s older names, short for “block make”) at the keyboard.

Notice the Block Definition dialog box:



- Give the block a name in the **Name** field. You can enter nearly anything, from a single letter to 255 characters. Not permitted are the following characters: < > / \ " ' : ; ? * | = and space.

When you try to give the block a name that already exists in the current drawing, AutoCAD warns you:

Blockname is already defined. Do you want to redefine it?

Most times you enter **No**, and then change the name.



Notes Sometimes you deliberately want to redefine an existing block. Perhaps you made an error in creating the block or left out attribute definitions. In this case, answer **Yes** to the “Do you want to redefine it?” question.

If you need help seeing the names of blocks already defined in the drawing, click the down arrow in the **Name** field. AutoCAD displays the names of blocks in the current drawing.

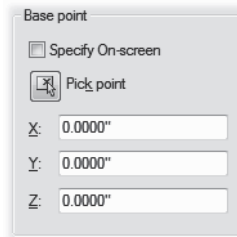
Block names that begin with * (asterisk), such as *X20, are created by AutoCAD. These are called *anonymous blocks*.

3. For the Base Point, enter coordinates in the **X**, **Y**, and **Z** fields, or else choose the **Pick Point** button to pick the insertion point in the drawing.

When you choose the **Pick Point** button, the dialog box disappears temporarily, and AutoCAD prompts you,

Specify insertion base point:

I recommend using an object snap mode to make the pick accurate, such as **ENDpoint** for the end of a line or **CENTER** for the center of a circle or arc.

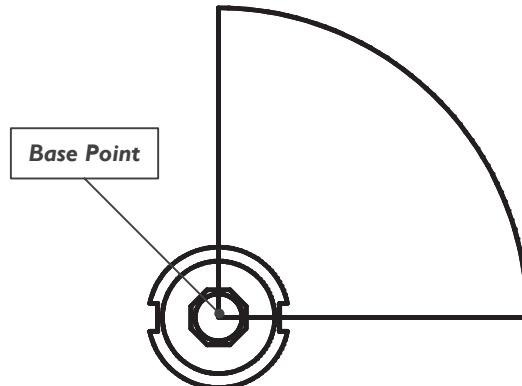


After you pick the insertion point, the dialog box reappears and AutoCAD fills in the **X**, **Y**, and **Z** fields. Normally, the **Z** value is 0, unless you have set the elevation or selected a point on a 3D object.



Note The base point defines the *insertion point*, which is where the block is later placed by the **INSERT** command.

The base point is usually at the lower left corner of the block for convenience, but sometimes is located elsewhere, such as the center of the block. In the figure below, the logical location for the sprinkler head's base point is the center of the head. That makes it easy to insert and attach it to existing objects, such as lines representing the irrigation piping.

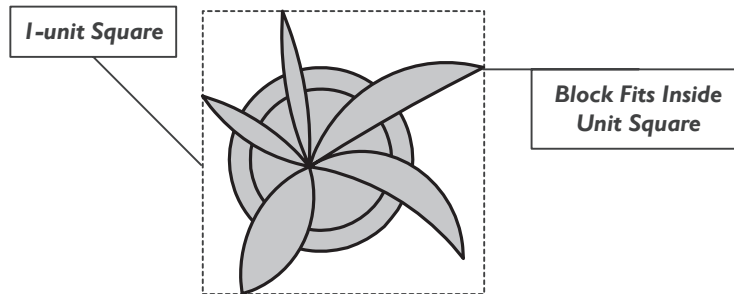


4. To select the objects that make up the block, choose the **Select Objects** button. The dialog box disappears and AutoCAD prompts you:
 Select objects: *(Select one or more objects.)*
 Select objects: *(Press ENTER to return to the dialog box.)*
 You can use any method of object selection, such as Window or Fence. After you finish selecting objects, press **ENTER** and the dialog box reappears. AutoCAD reports the number of objects you selected.
5. Click **OK**, and AutoCAD creates the block — even though it's hard to tell it happened. You access it later with the **INSERT** command.

Suggestions for Designing Blocks

When creating blocks, it is best to make them the size you intend for use. For example, office desks are commonly 24" x 36", so it makes sense to draw them that size.

Sometimes, however, you don't know what size will be used. For example, the block of a tree symbol could be inserted at 2' or 3' or 5' or whatever size you need. When you don't know how large the inserted block will be, draw the block at *unit size*. This means that the block fits inside a 1-unit square:



When it comes time to insert the block, the scale factor option sizes the block correctly and automatically.

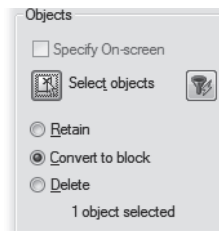
Blocks should be drawn on layer 0 so that they insert on the current layer. If they were created on a layer other than 0, say a layer named "Doors," then the blocks will always be inserted on layer Doors.

CREATING BLOCKS: ADDITIONAL METHODS

The Block Definition dialog box contains a number of options that help you create blocks.

Objects

The Objects section of the dialog box has these settings:



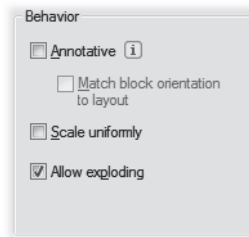
- ☐ **Specify On-screen** prompts you to select objects after you click the **OK** button to close the dialog box.
- **Select objects** clears the dialog box, and then prompts you to select objects in the drawing. After pressing **ENTER**, the dialog box returns.

- **Quick Select** displays the Quick Select dialog box, where you can select objects based on their properties; see the **QSELECT** command later in this chapter.
- **Retain** keeps the objects making up the block in the drawing as individual objects, as well as stores them in the new block definition.
- **Convert to Block** erases the objects, and replaces them with the block in the same position in the drawing (default).
- **Delete** erases the objects, while storing the block definition in the drawing. The block seems to disappear, which can freak out new AutoCAD users: “Wha’ happened to my drawing!!!!” You can insert the block with the **INSERT** command.

Objects Selected reports the number of objects selected for the block. You cannot create a block with no objects.

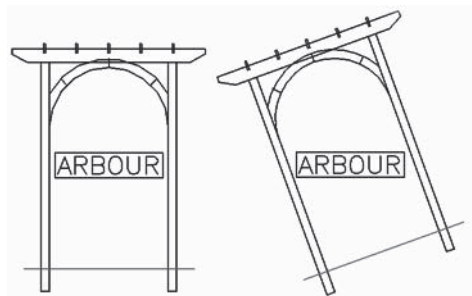
Behavior

The Behavior section determines how the block reacts during and after insertion.



- **Annotative** gives the block annotative scaling, which means it is displayed only when the viewport scaling matches the block’s annotative scale factor. (Clicking the **i** icon displays online help for this feature.)
- **Match Block Orientation to Layout** forces the block to remain “upright,” even when rotated with the **ROTATE** command or through layout rotation for plotting.

In the figure below, the boxed “Arbour” text is a block, with orientation turned on. When all objects in the drawing are rotated, the block remains in place.



- **Scale Uniformly** fixes the scale factor ratio at 1:1:1. This means the block cannot be inserted with different scale factors in the x, y, and/or z directions. It’s a good idea to turn on this option when you need the block to remain proportional.
- ☑ **Allow Exploding** allows the block to be reduced to its constituent parts with the **EXPLODE** command. Exploding causes block insertions to sever their link with block definitions, and to lose attribute data, if any. To prevent the block from being exploded, turn off this option.

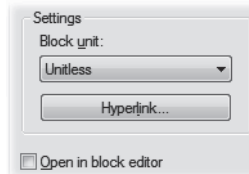
Settings

The Settings section sets settings.

- **Block Unit** specifies units for the block. Usually, AutoCAD makes one drawing unit equal to the current unit.

When you drag blocks from DesignCenter or *i-Drop*-enabled Web sites into drawings, AutoCAD automatically scales the block. (*i-Drop* is Autodesk technology that allows you to drag blocks from Web sites directly into drawings.)

Select a drawing unit from the droplist:



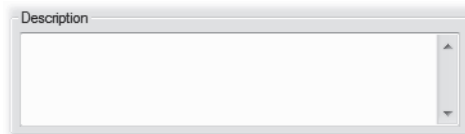
Inches	Nanometers	Feet	Microns
Miles	Decimeters	Millimeters	Decameters
Centimeters	Hectometers	Meters	Gigameters
Kilometers	Astronomical Units	Microinches	Light Years
Mils	Parsecs	Yards	Angstroms
Unitless			

(1 parsec = 3.26 light years; 1 astronomical unit = average distance between Earth and Sun; 1 angstrom = typical size of an atom, or 1 ten-billionth of a meter; 1 microinch = 1 millionth of an inch.)

- **Hyperlink** displays the Insert Hyperlink dialog box for attaching hyperlinks to blocks.
- ☐ **Open in Block Editor** opens blocks in the Block Editor after you click **OK**. This allows you to edit blocks further, or to turn them into a dynamic blocks.

Description

The description text is displayed by the INSERT and DESIGNCENTER commands.

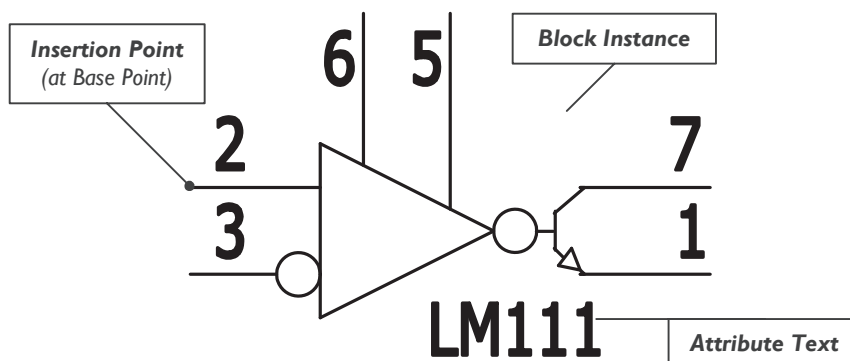


- **Description** provides room for you to include a long (or short) description of the block. This item is completely optional.

INSERTING BLOCKS

After you define blocks with the BLOCK command or copy them from other drawings, you can place them with the INSERT, DESIGNCENTER, or TOOLPALETTE commands.

INSERT needs at least two pieces of information to place blocks: (1) the name, and (2) the insertion point. In addition, you can optionally specify the scale factors, normally 1.0; the rotation angle, usually 0 degrees; and other options. (DESIGNCENTER and TOOLPALETTE need only the insertion point.)



The *insertion point* is the x, y-coordinate in the drawing where the block is inserted, as illustrated below. More specifically, the block is inserted at its base point, which was defined early during the block's creation. The base point becomes the insertion point.

After blocks are inserted, you can use the INSeRTion object snap to snap to their insertion points.



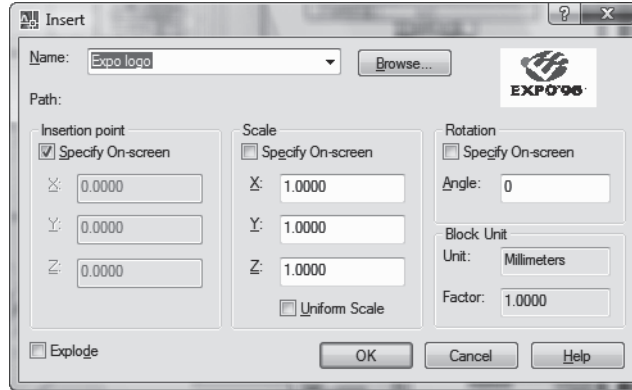
Note The inserted block resides on the layer that is current when the block is placed. Objects within the block retain the color, linetype, and so on of their original layer (i.e., the layer they were on before they were made part of the block). The properties are called “ByBlock.”

There is one exception: if objects were on layer 0, then they take on the properties of the layer on which they are inserted.

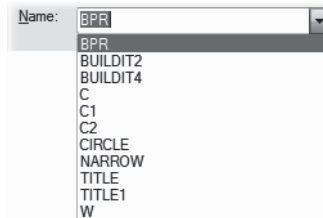
TUTORIAL: PLACING BLOCKS

1. To place blocks in drawings with the **INSERT** command, start the command with one of these methods:
 - In the ribbon's 2D Home panel, choose the **Insert** button from the Block panel.
 - Or, at the 'Command:' prompt, enter the **insert** command:
 Command: **insert** (Press ENTER.)
 - Alternatively, enter the aliases **i**, **ddinsert** (the command's old name), or **inserturl** at the keyboard.

Notice the **Insert** dialog box.

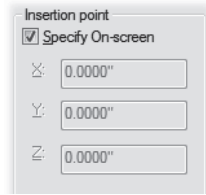


2. In order to insert a block, you select its name:
 - If the block resides in the drawing, enter its name in the **Name** field.
 - If you don't know the name of the block, click the arrow at the end of the **Name** field, and AutoCAD displays a list of all blocks defined in the drawing. Select one of them.



- If the block is not in the drawing, click **Browse** to select any .dwg drawing file to insert as a block. (Alternatively, you can use DesignCenter to search for and insert blocks and drawings.)
3. To specify the insertion point, you can enter the x, y, and z coordinates in the dialog box or in the drawing. The **Specify on Screen** option determines where this takes place:

- ☐ Specify the insertion point in the **X**, **Y**, and **Z** text entry boxes of the dialog box.
- ☒ Specify the insertion point in the drawing. After you click the **OK** button, AutoCAD prompts:
Specify insertion point or [Scale/X/Y/Z/Rotate]: (*Pick a point, enter an option, or type the x, y, z coordinates.*)



4. After you specify the insertion point, AutoCAD places the block in the drawing. Technically, this is called “inserting an instance of the block definition.”

INSERTING BLOCKS: ADDITIONAL METHODS

The Insert dialog box contains several options:

- **Scale** changes the size of the block instance.
- **Rotation** rotates the block instance.
- **Explode** inserts the block with its original components, not as an instance.

In addition, the command-line version of the command lists a number of options, including some that are hidden from you:

Insert Option	Meaning
Basepoint	Relocates the block’s base point.
Scale	Specifies a single scale factor for the x, y, and z-directions.
X	Scales the block in the x-direction.
Y	Scales the block in the y-direction.
Z	Scales the block in the z-direction.
Rotate	Rotates the block in the counterclockwise direction.

The following options are hidden:

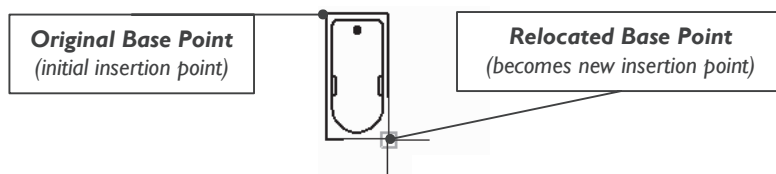
PScale	Sets the preview scale factor for the x, y, and z-directions.
PX	Sets the preview scale factor and insertion scale for the x-direction.
PY	Sets the preview scale factor and insertion scale for the y-direction.
PZ	Sets the preview scale factor and insertion scale for the z-direction.
PRotate	Sets the preview rotation angle and insertion scale factor.

Basepoint

The **Basepoint** option allows you to specify a new base point, the point at which the block is inserted. This can make it easier inserting blocks relative to other objects, such as placing a bathtub symbol into the corner of the bathroom. AutoCAD temporarily places the block in the drawing, and then prompts you:

Specify base point: (*Pick a point to relocate the base point.*)

Pick a point, and AutoCAD proceeds with the ‘Insertion point:’ prompt, allowing you to place the block in its intended location.

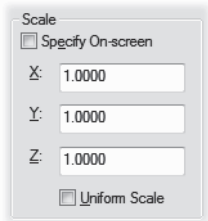


Changing the base point affects only this block insertion; it does not affect the block reference's base point. In other words, you need to reuse this option every time you insert the block and need a different base point.

Scale

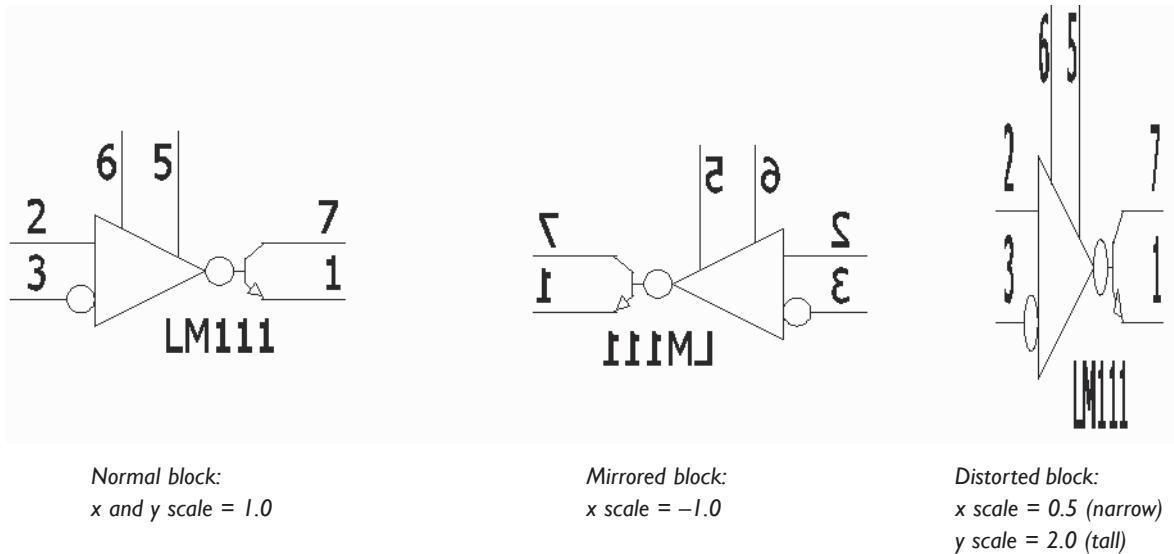
AutoCAD normally uses a *unit* scale factor for the inserted block. (“Unit” means that the scale factor is 1.0, so that the block is inserted at its original size.) There are times, however, when you might want the block to be a different size.

When the scale factor is more than 1.0, the block is made larger; when the scale factor is less than 1.0, the block is smaller.



The three scale factors — X, Y, and Z — can be the same or different, negative or positive. (When the **Scale Uniformly** option is turned on during block creation, only the X Scale factor is available.) When the scale factors are different from each other, the block is stretched; when negative, the block is mirrored, including its text; when positive, the block is inserted the right way around.

The illustration below shows the effect of some different scale factors on inserted blocks:



The **Specify On-screen** option determines whether you specify the scale factor in the dialog box or in the drawing:

- ☐ Specify the scale factor in the X, Y, and Z text entry boxes.
- ☒ Specify the factor in the drawing. After you click the Insert dialog box's OK button, AutoCAD will prompt you:

Enter X scale factor, specify opposite corner, or [Corner/XYZ] <1>: (Press ENTER to accept the default factor, enter a different x-scale factor, pick a point, or specify an option.)

Enter Y scale factor <use X scale factor>: (Press ENTER to make the y scale factor the same as x, or enter the y-scale factor.)

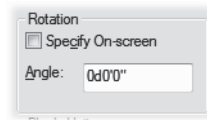
AutoCAD presents a large number of options at the command line. Here's what they mean:

Scale Option	Meaning
X scale factor	Scales the block in the x-direction.
Opposite corner	Specifies a rectangle that defines the x and y scale factors; insertion point is the first corner.
Corner	Same effect as the above option.
XYZ	Scales the block independently in the x, y, and z-directions.
Y scale factor	Scales the block in the y-direction.
Use X scale factor	Makes the y scale factor equal to the x scale factor.

The **Uniform Scale** option makes the y and z scale factors the same as the x factor.

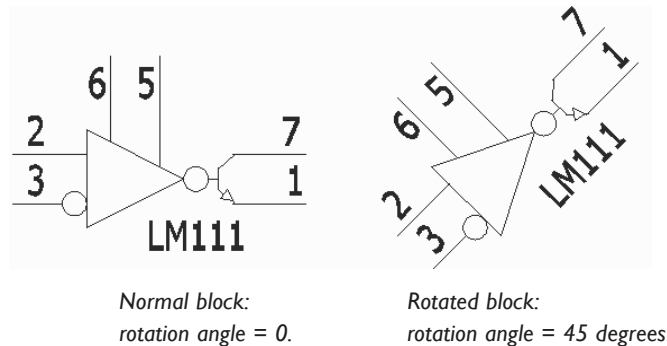
Rotation Angle

You can have AutoCAD rotate the block about its insertion point relative to the original orientation of the block. The **Specify On-screen** option determines when the specification takes place:



- ☐ Specify the rotation angle in the **Angle** text entry box.
- ☒ Specify the angle in the drawing. After you click **OK**, AutoCAD prompts you:

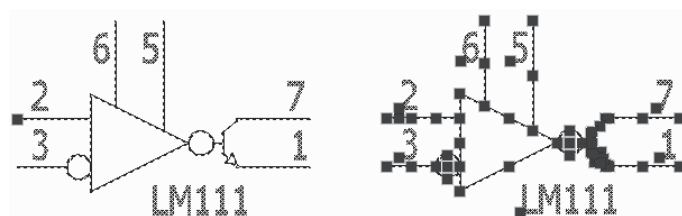
Specify rotation angle <0>: (Enter the rotation angle, or pick two points to indicate the angle.)



If you choose to indicate the angle, move the cursor. AutoCAD ghosts a rubber band cursor between the previously-set insertion point and the cursor. Move the cursor until the rubber-banded line shows the desired angle, and then click. The distance between the two points is irrelevant. The angle, shown by rubber-banded line between the insertion point and the angle point, determines the angle of insertion.

Explode

When you turn on the **Explode** option, AutoCAD inserts the block's constituent parts in the drawing — the lines, arcs, and text that make up the block definition.



Normal block: a single grip.
Exploded block: grips for every constituent part.

This option is not available when the **Allow Exploding** option was turned off during block creation. Note that AutoCAD limits you to specifying a single scale factor for exploded blocks, which applies equally to the x, y, and z directions.

Preview

The Preview window shows you what the block looks like.



Icons in the preview window indicate special blocks:

-  dynamic blocks.
-  annotatively-scaled blocks.

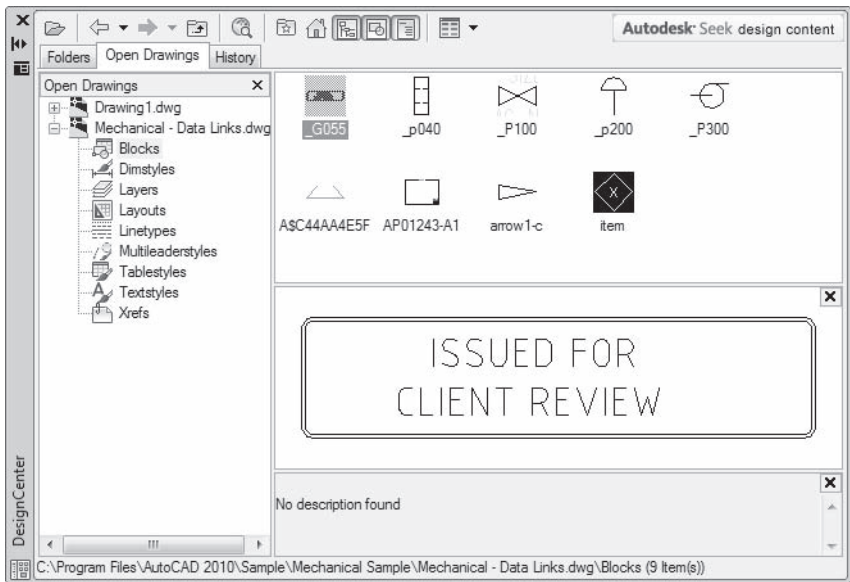
 DESIGNCENTER AND **TOOL PALETTES**

The drawback to the **INSERT** command is that it provides a preview of just one block at a time. Oft times, it is easier to pick out blocks visually from a larger group based on their look, rather than by their names. Other times, you may not be sure of the block's location: which drawing or folder is it stored in? For these reasons, you may find it easier to use Design Center or Tool palettes to insert blocks, which let you see groups of blocks, as well as use drag'n drop insertion.

(This section describes how blocks work with Design Center and Tool palettes; Appendix F provides greater details of these two commands.)

Inserting Blocks with DesignCenter

DesignCenter displays all block definitions in the current drawing, as well as in blocks in drawings stored on your computer, on the local network, and the Internet.



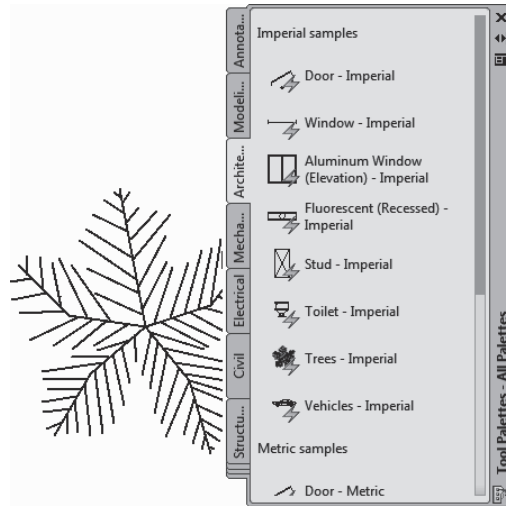
After you find the block you want, you simply drag it into your drawing. There are no prompts to answer, because AutoCAD determines the scale factor automatically, and determines the insertion point by where you let go of the mouse button. Object snaps help you place blocks accurately.

When you need more control, right-click the block's icon in DesignCenter, and then select **Insert Block** from the shortcut menu. AutoCAD displays prompts on the command line for setting insertion point, scale factor, and rotation angle options before the block is inserted in the drawing.

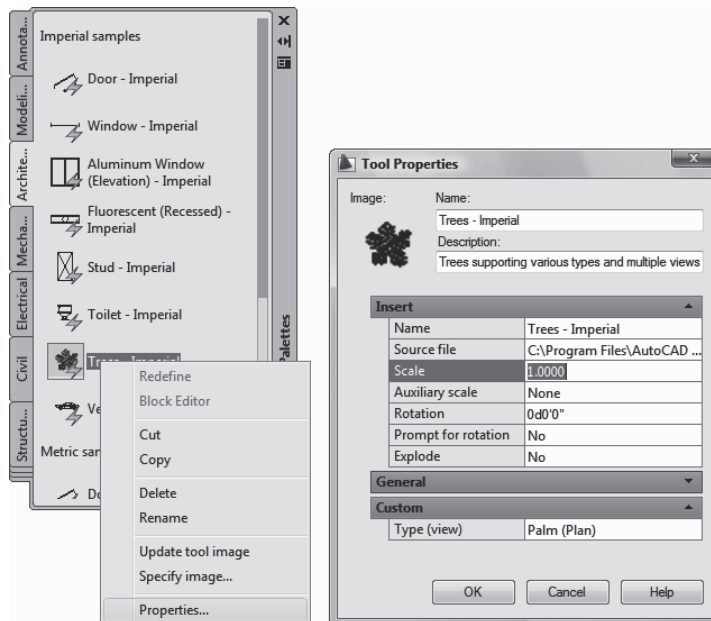
Inserting Blocks with Tool Palettes

Whereas DesignCenter displays all blocks definitions in drawings automatically, the Tool palette is meant to contain the blocks you use the most; you add them manually. You add blocks to the palette by dragging them from the drawing or DesignCenter onto a palette. Once on the palette, the blocks become available to any drawing you open or start.

Once on the palette, you drag blocks into drawings — just like DesignCenter.



When you need to control placement of a block, right-click its icon, and then select **Properties** from the shortcut menu. The Tool Properties dialog box lets you change the scale factor, rotation angle, and so on. These changes are semi-permanent: they affect future insertions of the block until you again change the properties.



(A drawback to Tool palettes is that they display images of blocks that no longer exist. Dragging the icon of non-existent blocks results in nothing.)



Note You can drag .dwg files from Windows Explorer into drawings. AutoCAD treats them differently, depending on where you drag them.

Drag files to the title bar — AutoCAD opens the .dwg files as drawings.

Drag files into the current viewport — AutoCAD inserts the .dwg files as blocks.

CREATING AND INSERTING BLOCKS: ADDITIONAL METHODS

AutoCAD has several commands that are useful for working with blocks:

- **BASE** command specifies the base point (origin) of the drawing.
- **WBLOCK** command exports blocks as .dwg files on disk.

Let's look at each command.

Base

The INSERT command can insert other drawings as blocks into the current drawing. The inserted drawing is normally inserted with coordinates 0,0 as its base point. If you wish to change the base point, use the BASE command on the drawing *before* inserting it.

Command: **base**

Specify base point <default>: *(Pick the point for the new base point.)*

Save the drawing to save the new base point, and then insert it in the other drawing.

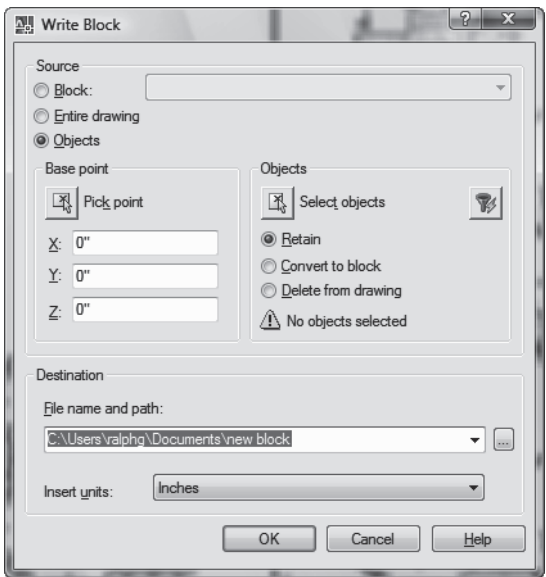
WBlock

The WBLOCK command is the complement of the INSERT command. Whereas INSERT inserts .dwg drawing files as blocks, WBLOCK saves all or part of the current drawing as another .dwg file on disk. This is an alternative to using the DesignCenter for sharing blocks.

- At the 'Command:' prompt, enter the **wblock** command.
- Alternatively, enter the **w** alias at the keyboard.

Command: **wblock** (Press ENTER.)

AutoCAD displays the Write Block dialog box, which looks similar to the BLOCK command's dialog box. The primary difference is in the Source area, where you choose what you want written to disk.



Source

Block saves blocks as *.dwg* files; the blocks must exist in the current drawing.

Entire Drawing saves the whole drawing with an exception: unused layers, blocks, styles, and dimension styles are not saved. This is a quick way to clean up a drawing.

Objects saves parts of the drawing; you specify the base point, and then select the objects.

Destination

In the **Destination** area, specify the file name and location (drive and folder.)

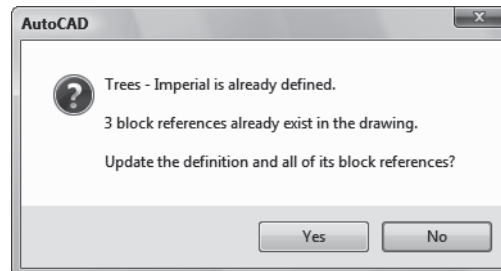
When you change the file name extension from *.dwg* to *.dxf*, AutoCAD saves the drawing or block in DXF format. DXF is short for “drawing interchange format.” (With AutoCAD 2004, Autodesk removed WBLOCK’s ability to save drawings in XML format — extended markup language.)

REDEFINING BLOCKS

After you insert the same block many times in drawings, you may need to change the block. By *redefining* one block, you can change all inserted blocks of the same definition. This is an especially powerful feature: imagine being able to change 100 identical drawing parts in a single operation!

Here’s how:

1. Explode one block that has been inserted. This action breaks the link to its block definition and reduces the block to its constituent parts.
2. Edit the exploded block.
3. Use the **BLOCK** command to convert the edited parts back into a block. Give this “new” block the same name. You may select the name from the **Name** droplist.
4. Click **OK**. AutoCAD displays the following dialog box:



5. Click **Yes**. Notice that all other blocks of the same name change, including those inserted with **MINsert**.

If the block was dynamic, all of its dynamism is lost when exploded.

If the block is another drawing that was inserted whole, edit the original drawing, and then reinsert it as described above.



BEDIT

The **BEDIT** command invokes the Block Editor environment for editing blocks, as well as making them dynamic (interactive).

The Block Editor is a separate editing environment. Many (but not all) AutoCAD commands are available for drawing and editing the blocks. In addition, the editor includes unique commands for constructing dynamic blocks.

The Block Editor performs these functions:

- Creates new blocks.
- Edits existing blocks.
- Adds dynamic actions to blocks.
- Converts dimensional constraints to parameter constraints (new to AutoCAD 2010).

Inside the editor, there are several special commands for all block definitions (those operating only inside the Block Editor). Additional system variables affect the display of dynamic blocks only while in the Block Editor.

TUTORIAL: EDITING BLOCKS WITH THE BLOCK EDITOR

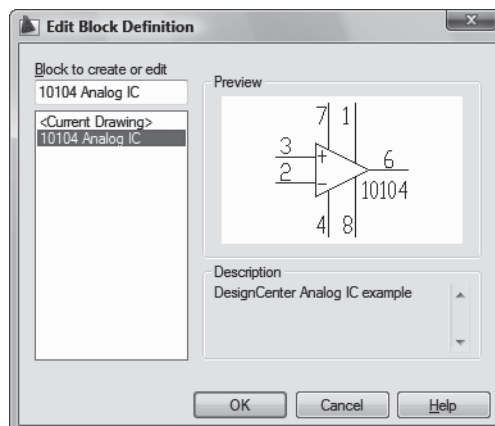
1. From the CD, open *bedit.dwg*, a drawing that contains a block.
2. To edit the block with the Block Editor, invoke the **BEDIT** command by one of these methods:
 - In the ribbon's 2D Home panel, choose the **Block Editor** button from the Block panel.
 - Or, at the 'Command:' prompt, enter the **bedit** command:

Command: **bedit** (Press ENTER.)
 - Alternatively, enter the **be** alias at the keyboard.



Note There are two alternate, faster methods to start the Block Editor: selecting a block, and then entering the **BEDIT** command opens the block in the Block Editor. Even faster is double-clicking the block, which opens it in the Block Editor.

3. (Click **Cancel** if AutoCAD asks, "Do you want to see how dynamic blocks are created?") Before entering the Block Editor, AutoCAD displays the Edit Block Definition dialog box.

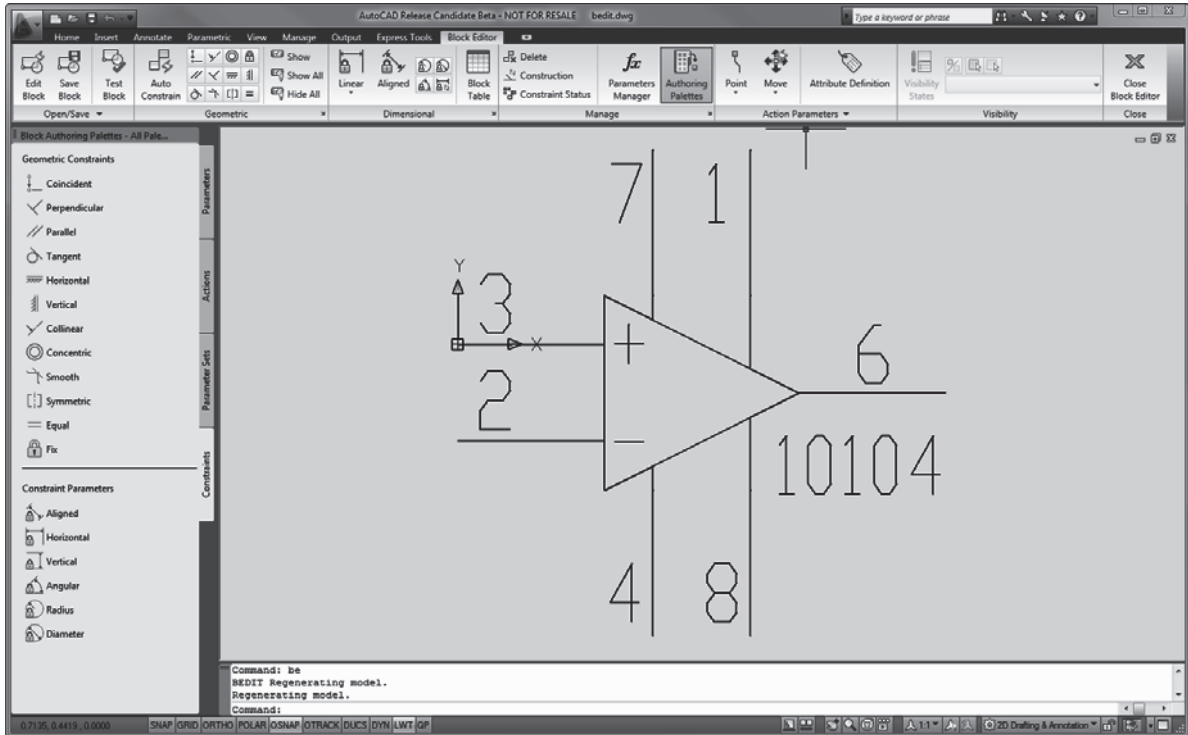


The dialog box provides you with three options:

- To edit an existing block, select its name from the list.
- To edit the entire drawing, select **<Current Drawing>**.
- To create a new block from scratch, enter a new name in the **Block to create or edit** field.

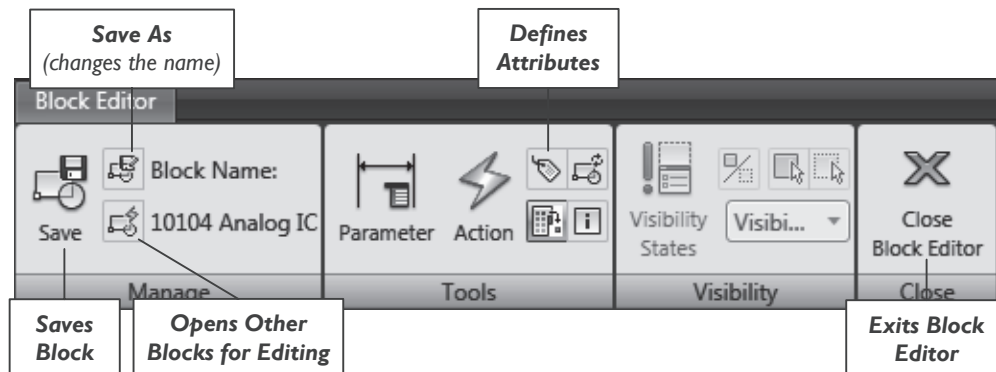
(When a yellow lightning icon appears in the corner of the preview window, this indicates a dynamic block is selected.)

4. Select a block to be edited. For this tutorial, select “10104 Analog IC,” and then click **OK**.



Notice the Block Editor. It looks much like regular AutoCAD, but the background color is pale gray, and it includes a ribbon and palette specific to block editing. In addition, the UCS icon is shifted to the block's insertion point.

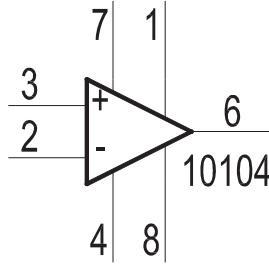
5. The palette is useful only for dynamic blocks, so close it with the **BAUTHORPALETTECLOSE** command, or click the **x** in its upper left corner.



The toolbar contains some commands useful for block editing, as illustrated below; the remaining commands are specific to editing dynamic blocks, and are not discussed here.

6. Make some changes to the block.

For instance, select the lines making up the triangle, and thicken them with the **Lineweights** droplist; use the **STYLE** command to change the font to Arial.



7. Use the **BSAVE** command to save the changes.
8. On the toolbar, click the **Close Block Editor** button.
Back in AutoCAD's drawing editor, notice that previous insertions of the block take on the changes you made.

HATCH

The **HATCH** command places hatch and fill patterns in drawings, while the **HATCHEDIT** command edits the patterns.

ABOUT HATCHING

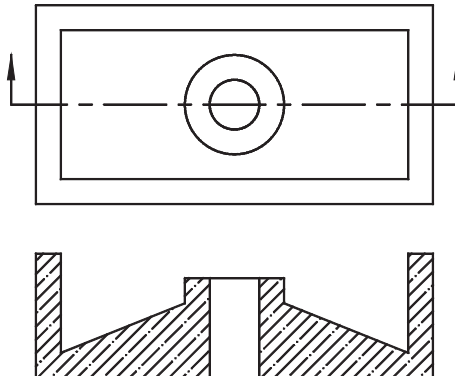
Sometimes it can be helpful to view objects as if they were cut apart, called a “section.” Mechanical designers use sections to show interior details of parts; civil engineers detail profiles by showing sections along roadways and railways; architects use sections through entire structures to show how buildings are to be constructed.

Hatching is used to show solid parts of sections, and AutoCAD easily hatches section views: after drawing a section, apply hatch patterns to cut areas with the **HATCH** command.

The spacing between hatch lines should be relative to the scale of the section, while the pattern should be oriented at 45 degrees from the main lines of the cut area, whenever possible.

Full Sections

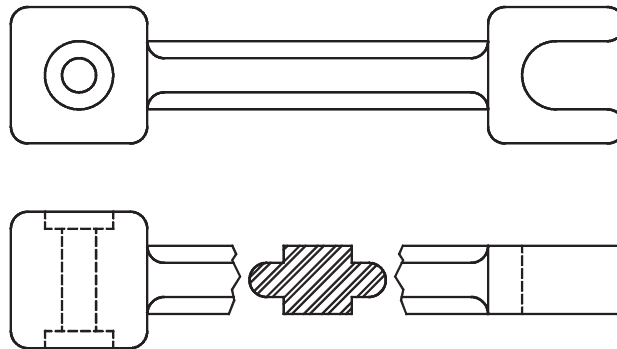
Full sections cut across the entire object, and are usually cut wherever the view needs to be clarified.



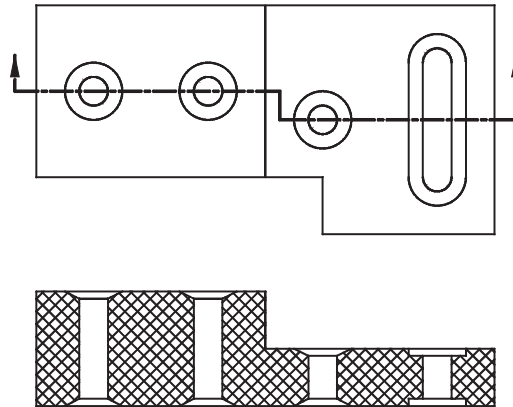
Parts that are “cut” are shown with crosshatching.

Revolved, Offset, and Removed Sections

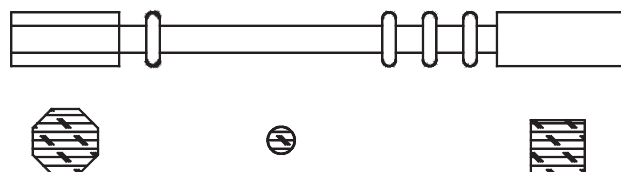
It can be helpful to view rotated sections of objects, to show their cross section more clearly. Such sections are referred to as *revolved sections*.



Offset sections are cut along an uneven line. Offset sections should be used carefully; change the cutting plane only to show essential elements.



Removed sections are similar to revolved sections, except the section is not placed at the point where the section was cut.



How Hatching Works

The **HATCH** command generates a boundary around an area automatically, and then places an associative hatch pattern within the area. *Associative hatch patterns* automatically update themselves when you change their boundaries, which is very helpful. If you prefer that the patterns not update themselves, then place non-associative hatches.

Hatches differ from other objects in that the lines making up the pattern are treated as one object. When you select one line, the entire hatch is selected.



To place hatch patterns, AutoCAD needs to know at least these parameters: (1) the name of the hatch pattern, (2) the area to be hatched, and (3) if it is associative. Optionally, you can also specify the scale, rotation angle, and style of boundary detection. You can also require that the boundary be retained, and you can create simple custom patterns.

BASIC TUTORIAL: PLACING HATCH PATTERNS

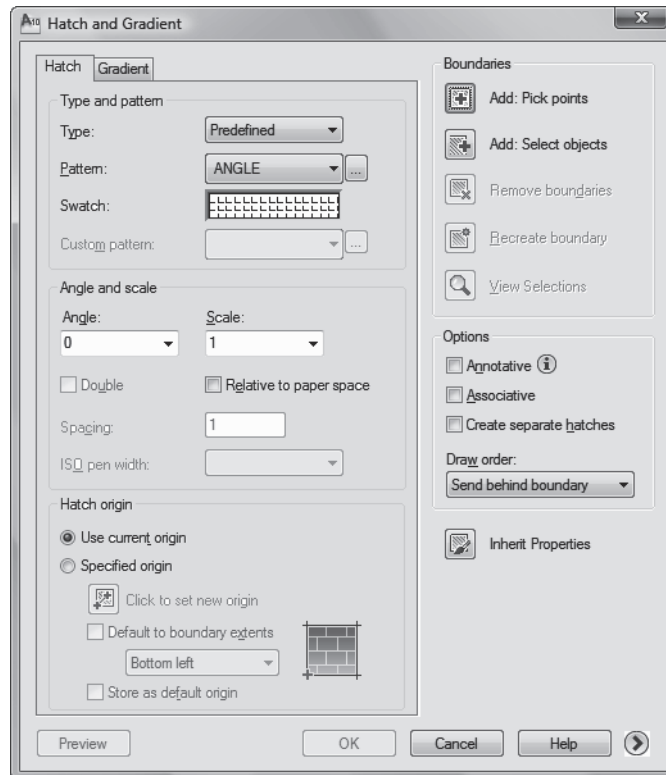
- I. To place hatch patterns in drawings, use the **HATCH** command with one of these methods:

- In the ribbon's 2D Home tab, choose the **Hatch** button from the Draw panel.
- Or, at the 'Command:' prompt, enter the **hatch** command:

Command: **hatch** (Press ENTER.)

- Alternatively, enter the aliases **h**, **bh**, or **bhatch** (one of its old names, short for "boundary hatch") at the keyboard.

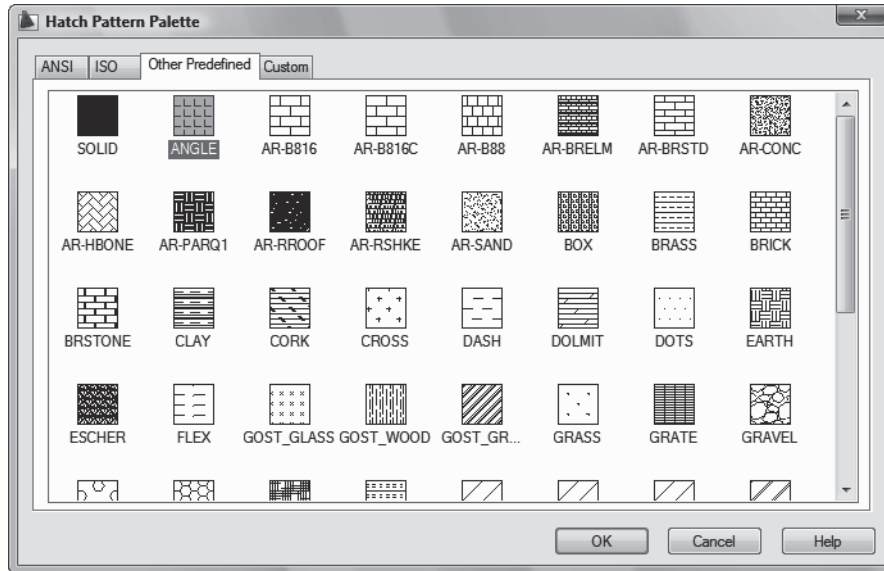
Notice the Hatch and Gradient dialog box.



2. From the Pattern droplist, select a hatch pattern name. If you prefer to select the pattern visually, click **Swatch** to display the Hatch Pattern Palette dialog box.

- **ANSI** — patterns defined by the ANSI (American) standard.
- **ISO** — patterns defined by the ISO (international) standard.
- **Other Predefined** — additional hatch patterns, ones that are the most useful for drawings.
- **Custom** — patterns defined by you.

For a solid color fill, select the "Solid" pattern found in the Other Predefined tab.



Select a pattern, and then click **OK**.

3. Select the area to be hatched by clicking the **Pick Points** button. Notice that the dialog box disappears, and that AutoCAD prompts you on the command line:

Pick internal point or [Select objects/remove Boundaries]: (Pick a point.)

Selecting everything... Selecting everything visible... Analyzing the selected data...
Analyzing internal islands...

Pick internal point or [Select objects/remove Boundaries]: (Press **ENTER** to return to dialog box.)

4. Click **OK**.

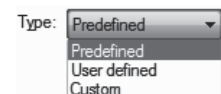
AutoCAD immediately hatches the area.

PLACING HATCH PATTERNS: ADDITIONAL METHODS

The Hatch and Gradient dialog box has two tabs, Hatch and Gradient, plus the **More Options** button that displays advanced options. This many options can be confusing, so let's step through them.

1. Select a Type of Hatch

The first step is to select the type of pattern. The **Type** droplist supports three classes of pattern.



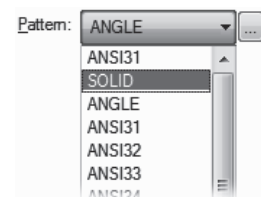
- **Predefined** selects a predefined hatch pattern listed by the Pattern droplist. Hatch patterns provided with AutoCAD are stored in the *acad.pat* and *acadiso.pat* files. (ISO patterns permit a pen width to be assigned to the pattern; "ISO" is short for the International Organization of Standards.)
- **User Defined** specifies a simple pattern of lines based on the angle and spacing. To create a unique pattern, apply linetypes to the hatching.
- **Custom** selects patterns in *.pat* files typically provided by third party or in-house developers.

Most of the time, you select a Predefined pattern.

2. Select a Pattern Name

Second, select a pattern from the **Pattern** droplist. The default pattern is one made of diagonal lines; it is named "ANSI31." To select a different one, click the down arrow, and chose one from the available hatch patterns.

If you don't recognize patterns by their name, the alternative is to click the **...** button. AutoCAD displays the tabbed dialog box of patterns illustrated above. Select a swatch, and choose **OK**.



3. Specify the Angle and Scale

Hatch patterns can be drawn at varying angles and scales. The sample swatches in the dialog box are shown at an angle of 0 degrees, even if the lines of the pattern are drawn at angles.

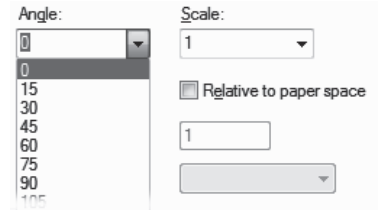
Select a preset angle in the Angle droplist, or else use the keyboard to enter a different angle.

Scale multiplies the hatch pattern by a factor. For instance, changing the value in **Scale** to 2 doubles the spacing between lines; changing it 0.5 reduces the spacing to half-size.

Select one of the preset scale factors in Scale box, or use the keyboard to enter another scale factor. Scaling hatch patterns is similar to scaling text and linetypes: enter the inverse scale factor in the Scale box. For example, if the drawing is to be plotted at a scale of 1:100, then the hatch pattern should be applied at a scale of 100. Better yet, click the **Annotative** button and the pattern scales itself.

When you turn on the **Relative to Paper Space** option, AutoCAD scales hatch patterns appropriately for each layout, using the layout's scale factor. (This option is grayed out in model space.)

To change the thickness of the lines, use lineweights.



Note The **OSNAPHATCH** system variable performs the useful function of determining whether object snap modes will snap to hatch patterns and gradient fills. When set to **0**, the default, then all osnap modes ignore hatch and fill patterns; when set to **1**, osnap modes select component objects within the hatch.

The **ISO pen width** option is available only when you select an ISO hatch pattern. It allows you to specify the width of the lines making up the ISO pattern.

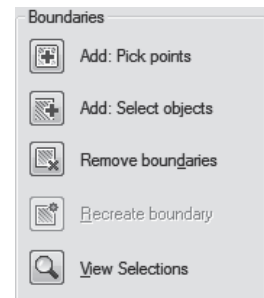
4. Selecting Hatch Boundaries

Hatches must be contained within *boundaries*; otherwise AutoCAD will not place the hatch. Boundaries are one or more objects that form closed polygons, and include objects like lines, polylines, circles, and arcs.

Before placing a hatch pattern, you identify the objects that form the closed boundary. If the boundary is not quite closed, you can use the **Gap Tolerance** option to tell AutoCAD to closeup the gaps temporarily (described later).

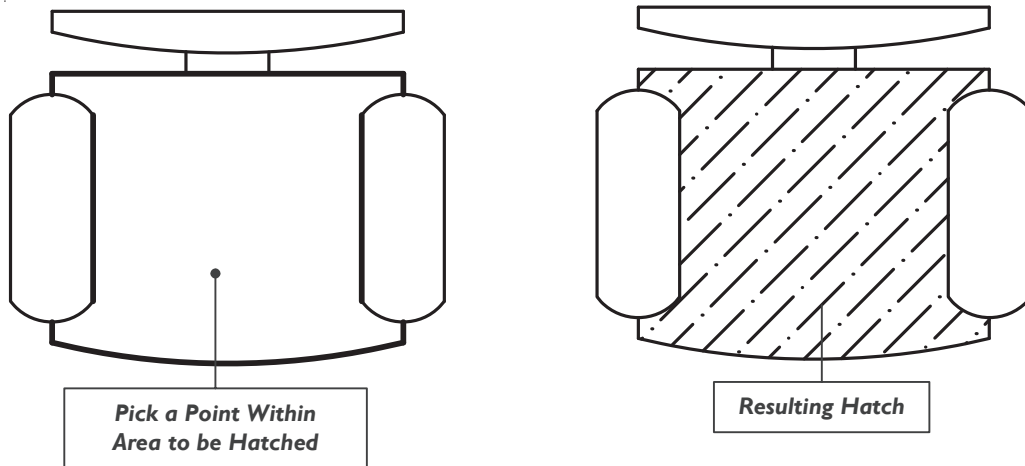
There are several ways by which you can identify the boundary:

- Pick points within areas to be hatched and let AutoCAD find the boundaries.
- Select the objects that make up the boundary.
- Remove islands from inside boundaries.



Defining Boundaries by Picking Areas

The simplest way to define the boundary is to show AutoCAD by selecting a point within the area to be hatched. Let's look at an example. The illustration shows an chair with areas that could be hatched: seat, back, arm rests.



In the Hatch and Gradient dialog box, choose the **Add: Pick Points** button. When the dialog box disappears from the screen, place the crosshairs in an area to be hatched, such as the seat.

Pick internal point or [Select objects/remove Boundaries]: (*Pick point.*)

AutoCAD analyzes the area, looking for leakage. Unknown to you, AutoCAD then places a polyline as the boundary around the inside of the area. (You can place boundaries by themselves with the **BOUNDARY** command, discussed later in this chapter.)

If you select a point in an area that is not contained, AutoCAD warns:

Valid hatch boundary not found.

When the boundary contains gaps, AutoCAD highlights them with red circles ([new to AutoCAD 2010](#)). Click **Close** to continue.

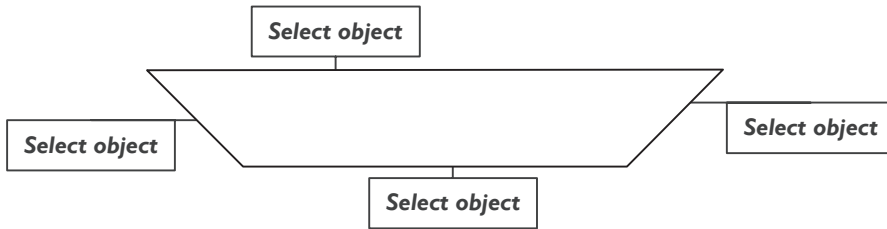


You must reconsider the area you want hatched; perhaps you need to draw another line to close off the area, or else define the maximum size of allowable gap.

Choose **Preview** to see how the hatch will turn out. When satisfied, choose **OK** and the hatch is completed.

Defining Boundaries by Selecting Objects

You can select the objects that will contain the hatch pattern. The illustration below shows a shape to be hatch that's been constructed with four lines.



To place a hatch in the polygon, choose the **Add: Select Objects** button. The dialog box disappears from the screen, and AutoCAD prompts you:

Select objects or [pick internal point/remove Boundaries]: *(Pick one or more objects.)*

Select objects or [pick internal point/remove Boundaries]: *(Press ENTER.)*

AutoCAD prompts you to select the objects that will form the boundary. Place the pickbox over each object, and then click.



Note Using the Window or Crossing option selects all the sides of the boundary in one operation.

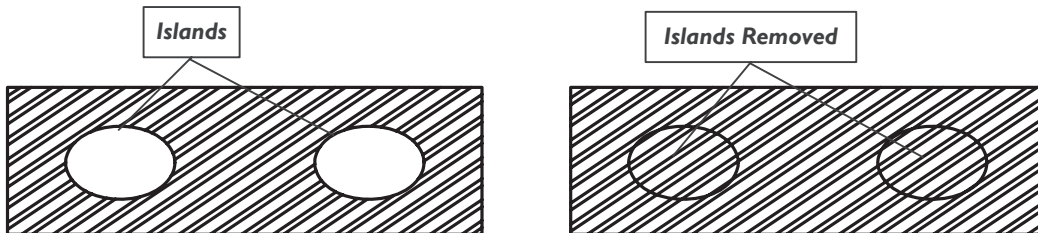
When you are finished selecting objects, press **ENTER**. The dialog box returns.

To test the boundary, choose the **Preview** button. Press **ENTER** or right-click to return to the dialog box.

To review the objects selected for the boundary, choose **View Selections**. The dialog box again disappears, and AutoCAD highlights the boundary objects. If you need to change the boundary objects, choose the **Select Objects** button to deselect and select objects.

Removing Boundaries

Sometimes the area to be hatched is not as straightforward as the examples hatched above. Hatch areas commonly to contain *islands*, other closed areas within the boundary — such as the two illustrated below.



AutoCAD initially assumes you do *not* want islands hatched. If, however, you want patterns drawn right through islands, then choose the **Remove Boundary** button. The dialog box disappears, and AutoCAD prompts you to pick the islands to be removed:

Select objects or [Add boundaries]: *(Pick object.)*

After you select an object, AutoCAD prompts you to pick another:

Select objects or [Add boundaries/Undo]: *(Pick another object.)*

Or, you can type **U** to deselect the object. When done removing islands, press **ENTER**. Back in the dialog box, choose the **Preview** button to see the effect of removing the islands.

Additional Options

The **Annotation** option turns on the annotative scaling property. When on, the hatch pattern does not appear in model space unless its scale factor matches that of the viewport. (Click the **i** button for online help for annotation.)

Associative

When AutoCAD draws hatch patterns, they are *associative*, which means they associate with their boundaries: change a boundary, and the hatch pattern updates by enlarging and shrinking itself to fit. AutoCAD draws the associative hatch pattern as a single object.

In some rare cases, you may want to place *non-associative* hatch patterns. This pattern is “dumb,” not knowing its boundary, or its parameters. AutoCAD draws the non-associative hatch pattern as a collection of independent lines collected into an anonymous block.

The **Associative** option lets you decide if the hatch pattern should be associative or not. If you are not sure, keep the option set to **Associative**, because you can later use the **EXPLODE** command to change associative hatch patterns to non-associative; going the opposite direction is not possible. When you select two or more boundaries, AutoCAD can fill them with one hatch, or each with its own.

The **Create Separate Hatches** option determines the number of patterns created when more than one boundary is selected: one for all, or one for each.

Draw Order

Hatch patterns and filled areas can overwhelm the drawing when they are displayed over top of other objects. This occurs when the hatches are placed later in the drawing process. The **Draw Order** option determines whether hatch patterns are displayed in front of or behind their boundary and other objects.

One solution is to freeze the layer containing the pattern; a better solution is to select **Send to Back**, which ensures the hatch patterns are visually underneath all other objects in the drawing.

This option is particularly useful for areas filled with solid color or gradients. You can use this option, together with the **DRAWORDER** and **TEXTTOFRONT** commands (described later), to control the display order of all objects in drawings.

Inherit Properties

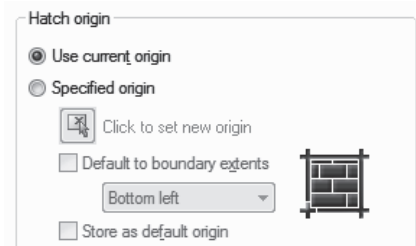
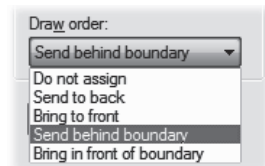
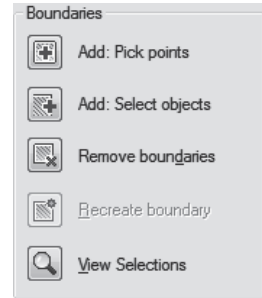
Associative hatch patterns are also self-aware. They know their parameters, such as pattern name, scale, and origin. Click the **Inherit Properties** button to copy the properties from an existing hatch pattern:

Select hatch object: (Pick another hatch pattern.)

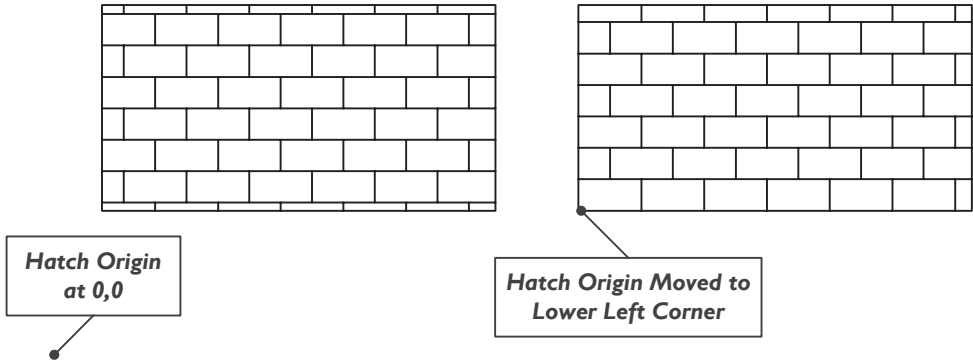
The dialog box then copies the properties for use with the current pattern.

Hatch Origin

The default origin of hatch patterns is 0,0 — coinciding with the origin of drawings. In many cases, that's fine, but sometimes you want adjacent hatches to line up with each other, or the pattern to begin at the edge of its boundary.



In the figure, the first hatch pattern has its origin at 0,0; notice that the brick pattern does not begin at the left or bottom edges. To correct this problem, the origin was moved to the lower left corner of the wall, and now the bricks line up correctly.



The **Hatch Origin** option allows you to relocate the origin, which is applied to all subsequent hatches — until you change the origin again. (Prior to AutoCAD 2006, the hatch origin was relocated using the SNAPBASE system variable.)

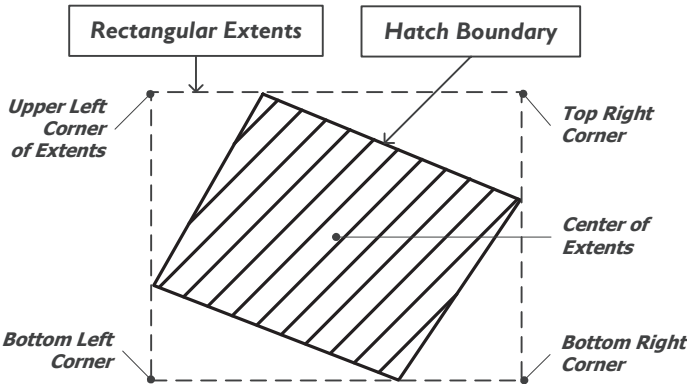
The **Use Current Origin** option reads the value stored in the HPORIGIN system variable, which is 0,0 until it is changed.

The **Specified Origin** option allows you to specify a different starting point for the hatch pattern:

Click to set new origin — click the button to pick a point in the drawing to relocate the origin of the hatch pattern.

Default to boundary extents — select one of the points on the extents of the pattern boundary: lower left, lower right, upper left, upper right, or center.

The *extents of the pattern boundary* is an imaginary rectangle that encompasses the hatch object. When the boundary is rectangular, then the extents match up; otherwise, they do not, as illustrated by the figure.



Advanced Hatch Options

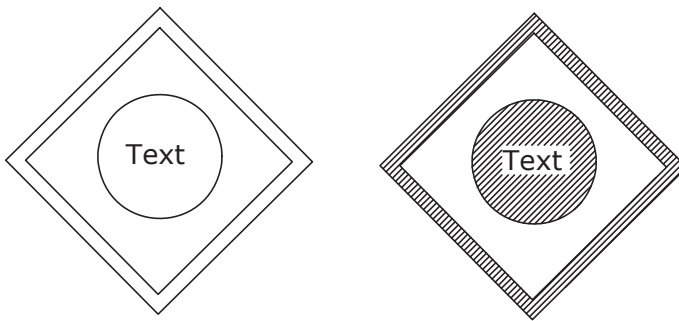
As you have read, the **Add: Pick Points** button effectively automates the process of finding the hatch area. In some cases, the hatch area might be too complicated for AutoCAD to determine automatically.

AutoCAD keeps hidden a collection of advanced (rarely used) options that are revealed when you click the **More Options** button. These options were formerly on the dialog box's Advanced tab.

Islands

When the area to be hatched contains other objects, the **Island Detection** option lets you choose which get hatched.

Let's look at how island detection works. The figure below illustrates a part with text.

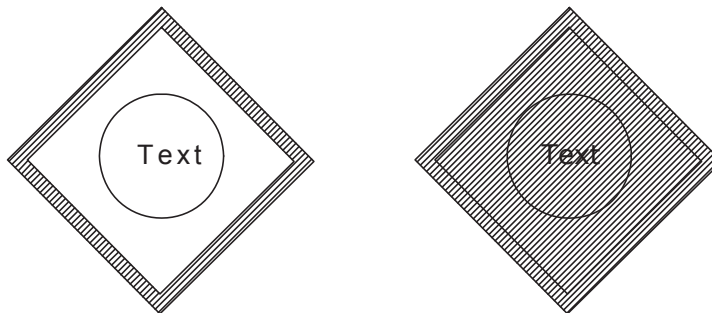


Left: Objects before being hatched.

Right: Islands and text hatched using Normal style.

Normal is the default detection style, which hatches inward from the outermost boundary, skips the next boundary, and hatches the next. In the figure above, notice how the text is bordered by the hatch. With this style, an invisible window protects text from being obscured by hatching.

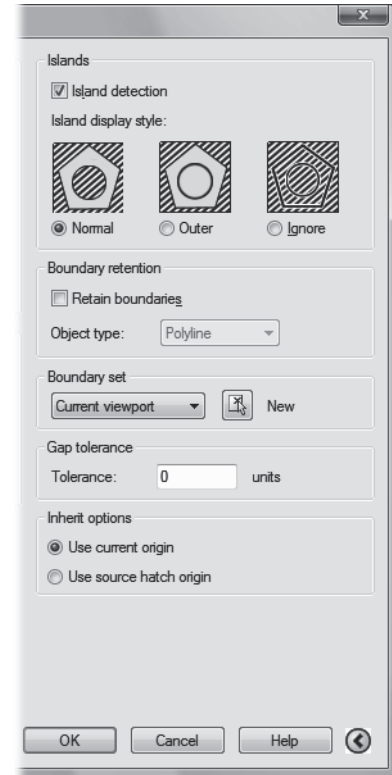
The **Outer** style hatches only the outermost enclosed boundary. The hatch continues only until it reaches the first inner boundary, and continues no further. Text is not hatched.



Left: Outer island hatched with Outer style.

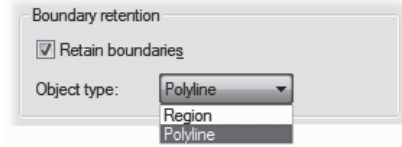
Right: Islands and text hatched using Ignore style.

The **Ignore** style hatches all areas defined by the outer boundary — with no exceptions. This style even hatches through text.



Boundary Retention

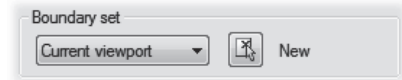
Boundaries are drawn as polylines or region objects. AutoCAD normally erases the boundary after the **HATCH** command is finished. To retain the polyline boundary in the drawing, check the **Retain Boundaries** box.



When the **Retain Boundaries** option is on, you can request that AutoCAD use a polyline for compatibility with older versions of AutoCAD; otherwise, use the **Region** option.

Boundary Set

When you use the **Add: Pick Points** option to define boundaries, AutoCAD analyzes all objects visible in the current viewport. You can, however, change the set of objects AutoCAD examines. In large drawings, reducing the set lets AutoCAD operate faster.



The **Current Viewport** option is the default, and examines all objects visible in the current viewport to create the boundary.

The **New** button prompts you to select objects from which to create the boundary set. (AutoCAD includes only objects that can be hatched.) The **Existing Set** option creates the boundary from the objects selected with the **New** option. (You must use the **New** option *before* the **Existing Set** option.)

Gap Tolerance

In the early days of AutoCAD, if a boundary had a gap, the hatch pattern could “leak” out, covering much of the rest of the drawing. Drawing hatches is a processing-intensive operation, and leaky hatches would bog down the slow computers of the day. Hence, CAD operators were careful to ensure hatches would not leak.

In more recent releases of AutoCAD, Autodesk programmers wrote code that checked for leaks before applying the hatching. If it found a gap, AutoCAD would warn, “Valid hatch boundary not found.” But CAD operators still had to fix gaps manually.

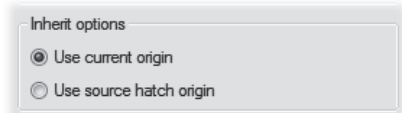
As of AutoCAD 2005, the **Gap Tolerance** option allows gaps in the boundary, ranging from 0 (the default value) to 5000 units. (*Warning!* Hatching to a gapped boundary leaves associative hatching turned off until you turn it back on again.)



Note Hatches can be handled more easily if they are put on their own layer. They can also be turned off and frozen to speed redraw time. Be sure that the layer linetype is continuous. Although the hatch pattern may contain dashed lines and dots, the linetype should be continuous to ensure a proper hatch.

Inherit Options

The **Inherit Options** option determines whether the origin is considered when properties are inherited from another hatch pattern.



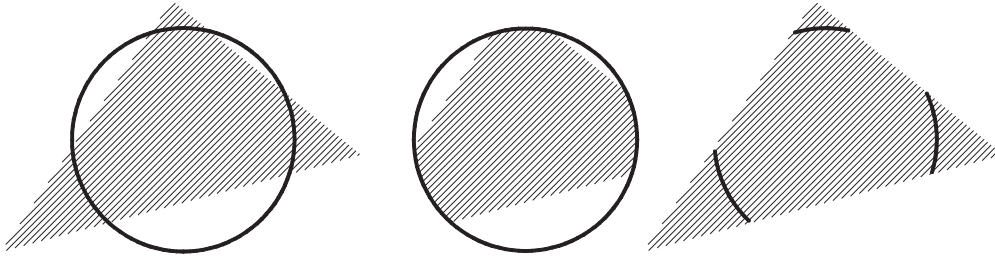
Use **Current Origin** ignores the origin of the source pattern.

Use **Source Hatch Origin** copies the origin from the source pattern.



Notes You can use the **TRIM** command to trim hatch and fill patterns. The process is similar to trimming other objects. (See Chapter 9, “Additional Editing Options.”) Start the **TRIM** command, and then select an object as the cutting edge. At the “Select object to trim” prompt, pick the portion of the hatch you want trimmed (removed).

Conversely, the hatch pattern can be used to trim other objects. Trimmed hatches are associated with the new boundary.



Left: Circle and hatch patterns — before trimming.

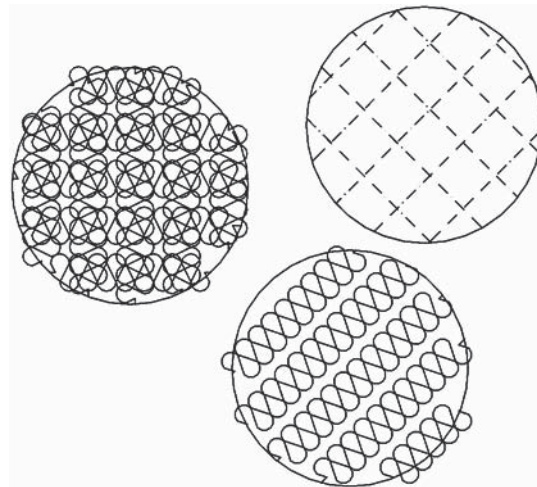
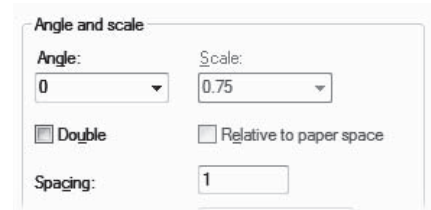
Center: The circle trimming the hatch.

Right: The hatch boundary trimming the circle.

Specifying User Defined Patterns

When you select **User Defined** as the hatch type, you work with a single pattern consisting of simple, parallel lines. You specify three parameters: angle, spacing between lines, and single or double.

The **Angle** option is the same as described earlier. Turning on the **Double** check box causes AutoCAD to draw the pattern twice, the second time at 90 degrees to the first pattern, creating a crosshatch. The **Spacing** option specifies how far apart the parallel lines are drawn. The illustration below shows examples of combining linetypes with user-defined hatch patterns.

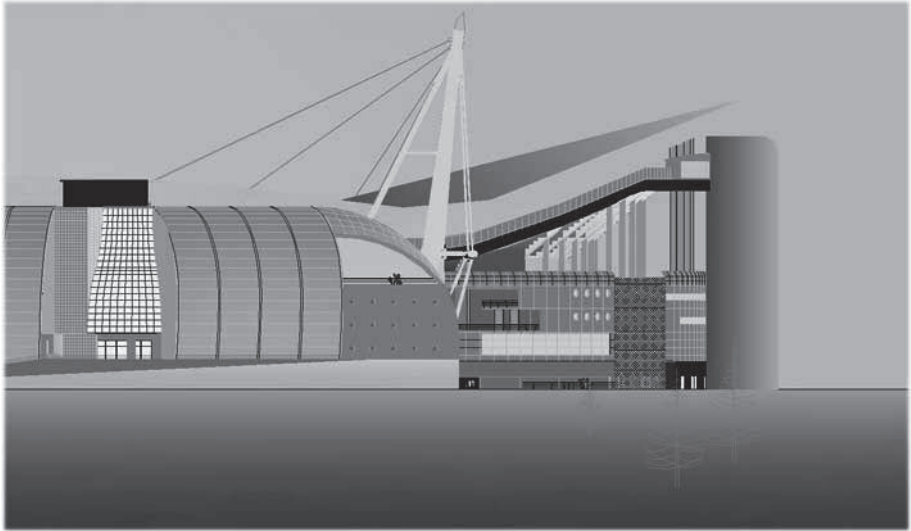


In the circle at left, I used the Batting pattern with double hatching; below it, with single hatching.

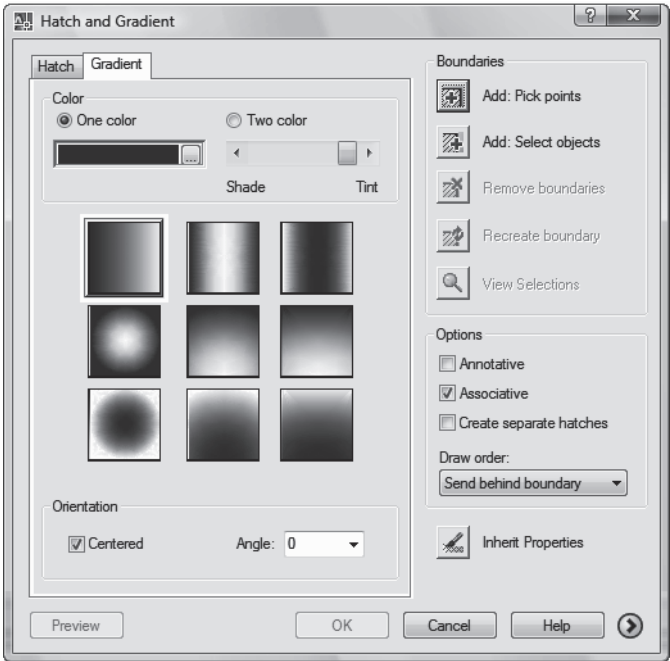
GRADIENT

The GRADIENT command places *gradient* fills.

Gradients are colors that change in intensity (more white or more black), or change from one color to another. Using gradients allows you to create 3D-like effects in 2D drawings, as the example below illustrates.

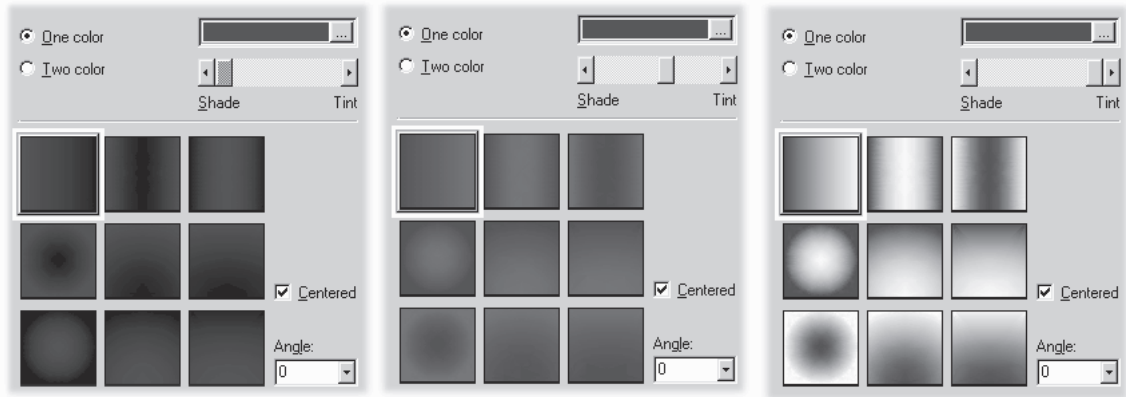


The GRADIENT command displays the Gradient tab of the Hatch and Gradient dialog box:



AutoCAD supports two types of gradients: one- or two-color. **One Color** means a single color is gradated to white or black. To change the base color, click the ... button, and then choose a new one from the Select Color dialog box.

To gradiate the color more towards black (Shade) or white (Tint), drag the **Shade-Tint** slider, as illustrated below.



Color gradients near black.

Color with little gradient.

Color gradients near white.

Two Color creates a gradient between two colors. Pick the ... button next to each color sample to select the colors. You don't have the slider to help you create subtle gradients; as a workaround, select two similar colors.

Centered toggles the gradient between being centered in the fill area, or starting at the left edge. **Angle** rotates the gradient inside the fill area.

HATCHEDIT

The HATCHEDIT command changes the properties of patterns and fills.

The command displays a dialog box that looks exactly the same as that of the HATCH command, but with some options grayed out, meaning they are unavailable.

TUTORIAL: CHANGING HATCH PATTERNS

- To change hatch patterns in drawings, use the **HATCHEDIT** command with one of these methods:
 - In the ribbon's 2D Home tab choose the **Edit Hatch** button from the Modify panel.
 - Double-click the associative hatch object.
 - At the 'Command:' prompt, enter the **hatchedit** command.
 - Alternatively, enter the **he** alias at the keyboard.

Command: **hatchedit** (Press ENTER.)
- AutoCAD prompts you to select the hatch pattern to modify:
 Select associative hatch object: (Pick a hatch pattern.)
 Notice the Hatch Edit dialog box.
- Make changes, such as to the pattern name or scale.
- Click **Preview** to see the effect of the changes. Notice that the dialog box disappears and that AutoCAD prompts you:
 Pick or press Esc to return to dialog or <Right-click to accept hatch>: (Enter an option.)
- To accept the changes, press **ENTER** or right-click the mouse.
 To return to the dialog box, press **ESC**, or left-click the mouse.

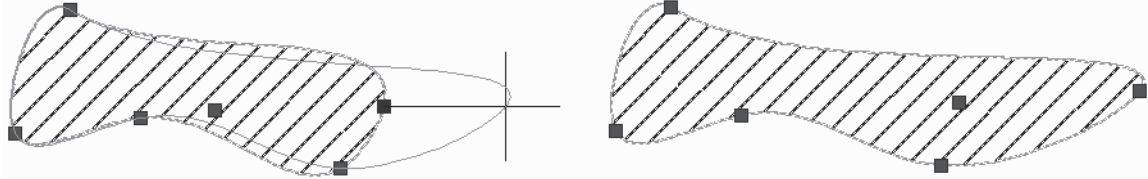
As an alternative, click the **Inherit Properties** button. This allows you to match the pattern to another in the drawing. AutoCAD temporarily dismisses the dialog box, and then prompts you:

Select associative hatch object: *(Pick a hatch pattern.)*

AutoCAD copies the properties of the other hatch pattern, such as its name, rotation angle, and scale factor. When the Hatch Edit dialog box returns, click **OK**. The pattern changes to match.

Editing Hatch Boundaries

The boundaries of hatch patterns can be stretched larger and smaller simply by dragging one of the grips (colored squares) found on the boundary, as illustrated below.



As of AutoCAD 2010, non-associative hatch patterns can be edited. Previously, non-associative hatches could only be moved. Now grips on the pattern's vertices can be dragged to create new shapes.

BOUNDARY

The **BOUNDARY** command creates single regions from disparate areas.

Earlier, you read how AutoCAD creates a temporary boundary out of a polyline or region to hold the hatching. To work with boundaries independent of hatch patterns, use the **BOUNDARY** command. The boundary is selected in the same manner, and is constructed as a polyline or region. The difference is that no hatch is placed within the boundary.



Note The **BOUNDARY** command converts the outlines of objects to polylines; this allows you to select them more easily for editing.

You probably won't notice the new boundary in the drawing, because AutoCAD traces over the objects defining the boundary. To edit it, for example to move or color it, use the **Last** object selection mode, which selects the last-drawn object in the drawing.

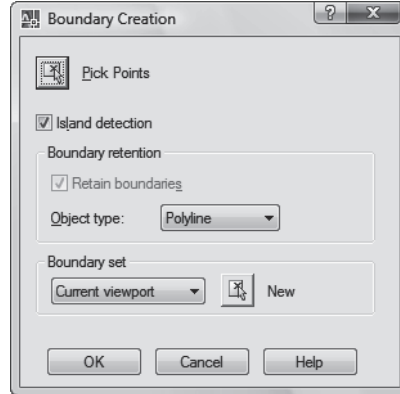
TUTORIAL: CREATING BOUNDARIES

1. To place boundaries in the drawing, use the **BOUNDARY** command with one of these methods:
 - In the ribbon's 2D Home panel, choose the **Boundary** button from the Draw panel.
 - At the 'Command:' prompt, enter the **boundary** command.
Command: **boundary** *(Press ENTER.)*
 - Alternatively, enter the aliases **bo** or **bpoly** (the command's original name, short for "boundary polyline") at the keyboard.

Notice the Boundary Creation dialog box.

2. Select the object you want used for the boundary: a polyline or a region.
Polyline boundaries are more easily edited, while region boundaries can be analyzed for properties, such as the centroid (geometrically-weighted center), using the **MASSPROP**

command (see Chapter 18).



3. Click **Pick Points** to pick a point within the area to be bounded. If the area is enclosed, AutoCAD creates the boundary, makes the following report, and exits the command:

BOUNDARY created 1 polyline

Command:

If AutoCAD finds the area is not closed, a dialog box complains in somewhat misleading terms, “Valid hatch boundary not found.”

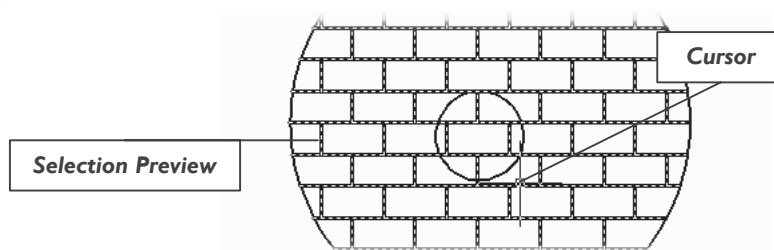
SELECT

The **SELECT** command selects objects in drawings. You typically select objects before editing them. For example, you can select all objects that are inside an area, or all objects in the entire drawing.

Most commonly, objects are not selected with **SELECT**, but during other commands that present the ‘Select objects:’ prompt. For example, you can start the **MOVE** command, and then select the objects you wish to move. Both the **SELECT** command and the ‘Select objects:’ prompt have identical options.

More usefully, AutoCAD allows you to select objects *without* this command or the prompt. When no command is active, you can pick one or more objects: AutoCAD highlights them, and displays *grips* as shown on the previous page. (These small squares are known as “handles” in other software application; in AutoCAD, handles are unique identifiers assigned to objects.) You then edit the object by manipulating its grips; press **ESC** to exit this direct editing mode.

As the cursor passes over them, objects take on highlighting and thicken their lines. When you click the mouse button, the object is selected. AutoCAD identifies selected objects by displaying them with *highlighting* — as if they were drawn with a dashed line.



Selection preview is controlled by options found in the Selection tab of the Options dialog box, described later in this chapter.

TUTORIAL: SELECTING OBJECTS BY LOCATION

1. To select objects in the drawing, use the **SELECT** command with one of these methods:
 - At the 'Command:' prompt, enter the **select** command.
 - Alternatively, enter a selection option at any "Select objects" prompt.

Command: **select** (Press ENTER.)
2. AutoCAD prompts you to select objects:

Select objects: (Select an object.)

1 found Select objects:

Notice that AutoCAD highlights the objects you select, and keeps a running tally of the number of objects selected, as in "1 found."
3. You can keep selecting objects until you press **ENTER** to exit the command:

Select objects: (Select more objects.)

3 found Select objects: (Press ENTER.)
4. After pressing **ENTER** to exit the **SELECT** command, the highlighting disappears. The objects you selected are added to AutoCAD's *selection set*.

So, you may wonder, what good is the **SELECT** command? Not a lot, which is why many CAD operators never use it. You can, however, access selection sets with the **Previous** option of the **SELECT** command, or at any 'Select objects:' prompt. In this way, the **SELECT** command is good for creating a selection set used later by other commands.



Note **SELECT** is one of the rare commands that does not list its options. To force it to do so, enter **?**, as follows:

Command: **select**

Select objects: **?**

Invalid selection

Expects a point or Window/Last/Crossing/BOX/ALL/Fence/WPolygon/CPolygon/Group/

Add/Remove/Multiple/Previous/Undo/AUto/Single/SUbject/Object

Select objects: (Enter an option.)

Select Options

That probably seems like too many options for you! It is useful to know about all of them, but in practice you probably use just a half dozen — All, Previous, Last, Window, Crossing, and point are the ones I use most frequently, while the technical editor uses Auto the most. "Because," he says, "Auto is the most efficient. Most of the other options are left over from older versions of AutoCAD."

ALL (Select All Objects)

The **All** option selects all the objects in the drawing, except those on frozen and locked layers — most of the time. For some commands, such as **COPYCLIP**, the **All** option selects only those objects visible in the viewport; in model space, it selects all objects, visible or not.



Notes As a shortcut, you can press the **CTRL+A** keys, which selects all objects in the drawing (subject to the restrictions noted above). To select nearly all objects in the drawing, first select all objects with the **All** option, and then use the **Remove** (explanation follows) option to "deselect" the exceptions.

Select Option	Meaning
Expects a point	Selects one object under the cursor.
Window	Selects all objects fully within a rectangle defined by two points. Window mode is indicated by a solid rectangle filled with blue color.
Last	Selects the most recently created visible object.
Crossing	Selects all objects within and crossing a rectangle defined by two points. Crossing mode is indicated by a dashed rectangle filled with green color.
BOX	Selects all objects within and/or crossing a rectangle specified by two points. If the selection rectangle is picked from: • Right to left, performs Crossing selection. • Left to right, Window selection.
All	Selects all objects in the drawing, except those residing on frozen and locked layers.
Fence	Selects all objects crossing a selection line. The fence line can cross itself.
WPolygon	Selects all objects completely within a selection polygon, which can be any shape but cannot cross itself; AutoCAD closes polygon.
CPolygon	Selects all objects within and crossing a selection polygon, which can be any shape but cannot cross itself; AutoCAD closes polygon.
Group	Selects all objects comprising a named group.
Class	Selects object classes defined by add-on software, such as Autodesk Map.
Add	Switches selection mode to Add, after being in Remove mode. Objects selected by any means listed above are added to the selection set.
Remove	Switches selection mode to Remove. Objects selected by any means listed above are removed from the selection set. As an alternative, you can hold down the SHIFT key to remove objects from the selection set.
Multiple	Selects objects without highlighting them. Also selects two intersecting objects when the intersecting point is selected twice.
Previous	Adds objects that were previously selected to the selection set. The Previous selection set is ignored when you switch between model and paper space.
Undo	Removes the object most recently added to the selection set.
Auto	Selects objects by three methods. Picking a point in a blank area of the drawing starts Window or Crossing mode, depending on how you move the cursor: • Right to left performs Crossing selection. • Left to right performs Window selection. • Picking an object selects it.
Single	Selects the first object(s) picked, and then does not repeat the “Select objects:” prompt.
SUbject	Selects subobjects on 3D models: vertices, edges, and faces; equivalent to holding down the CTRL key during object selection.
Object	Exits subobject selection mode.

Pick (Select Single Object)

To select a single object, click on it with the cursor. There is no need to enter any options at the “Select objects:” prompt:

Select objects: (*Pick an object.*)

In this mode, AutoCAD selects a single object. To select another object, pick again; to “unselect” the object, hold down the **SHIFT** key. To select from two or more overlapping objects, follow this procedure:

1. Hold down the **CTRL** key while selecting an object. If the object overlaps another, AutoCAD displays <Cycle on> at the 'Command:' prompt.
2. Release the **CTRL** key.
3. Click repeatedly (cycle) until AutoCAD selects the object you want.
4. Press the **ENTER** key to exit cycle mode and return to the 'Select objects:' prompt.

W (Select Within a Window)

One of the most common forms of object selection is the **W** option (short for “window”). It places a rectangle around the objects to be selected. You define the rectangle by picking its opposite corners, as follows:

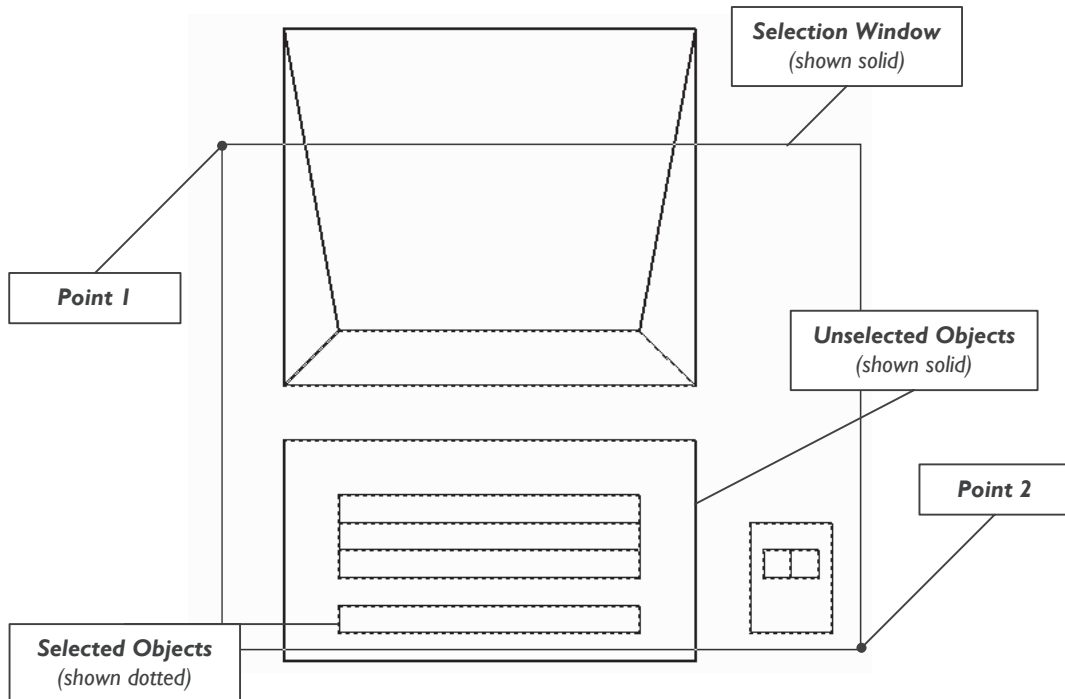
Select objects: **w**

Specify first corner: (Pick point 1.)

Specify opposite corner: (Pick point 2.)

Objects *entirely* within the window rectangle are selected. If an object crosses the rectangle, it is not selected (see **Crossing** mode). Only objects currently visible on the screen are selected.

You can place the selection rectangle so that all parts of those objects you want to choose are contained in the windows, while those that you don't want are not entirely contained. With this method, you select objects in an area of your drawing where objects overlap.



C (Select with a Crossing Window)

The **C** option (short for “crossing”) is similar to the **Window** option, with an important difference: objects crossing the rectangle are included in the selection set — as well as those objects entirely within the rectangle. Define the rectangle by picking its opposite corners.

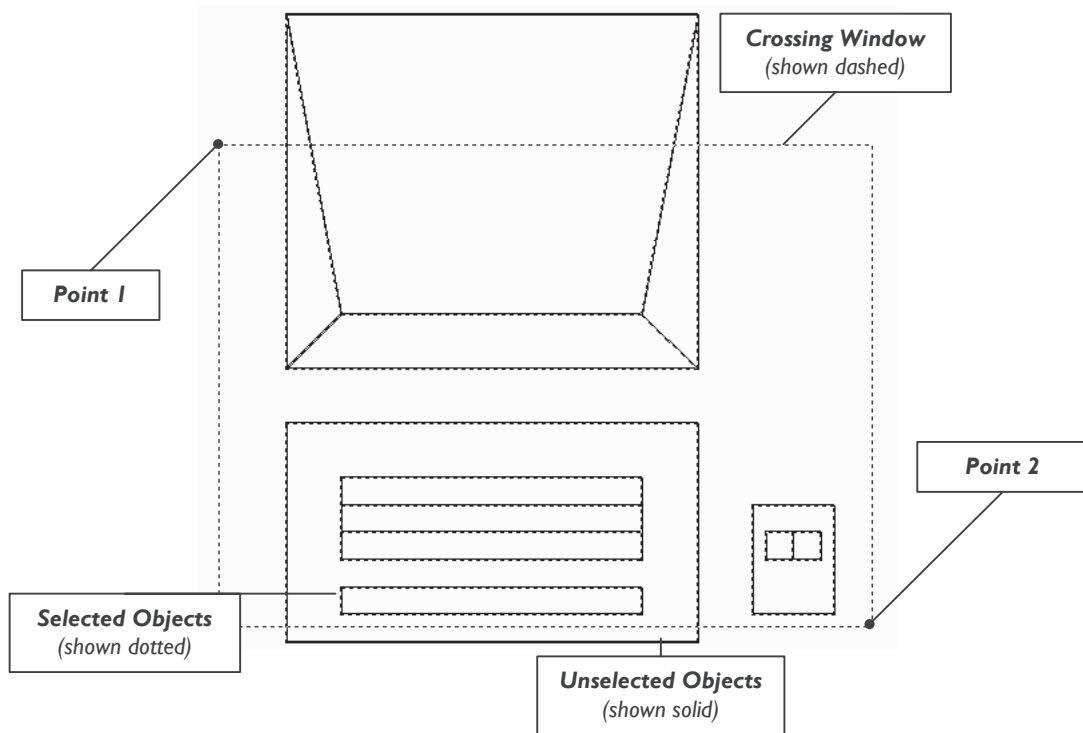
Select objects: **c**

Specify first corner: (Pick point 1.)

Specify opposite corner: (Pick point 2.)

Notice that the crossing window is dashed and filled with a transparent green color. This distinguishes it from the Window rectangle, which is made of solid lines and blue color.

The color and opacity of area selections can be changed with the Options dialog box: choose the Selection tab, and then click the **Visual Effect Settings** button.

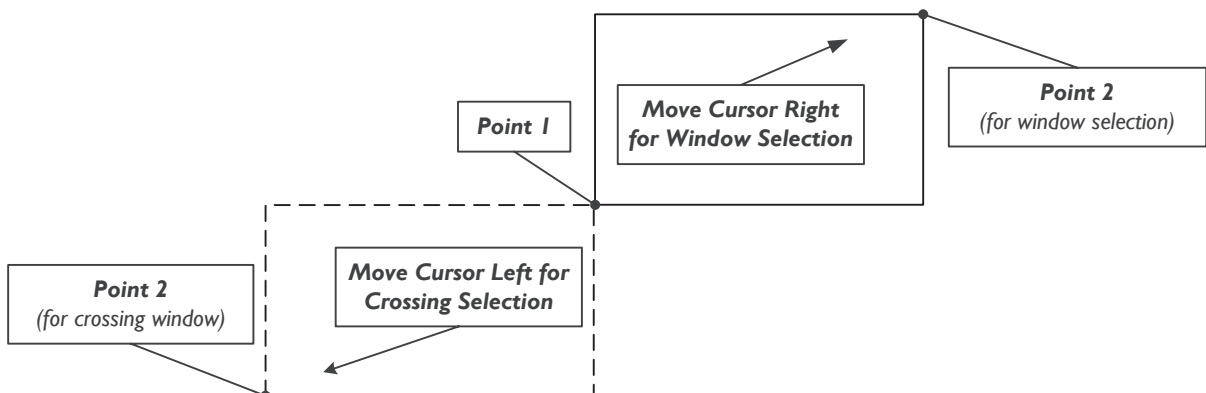


BOX (Select with a Box)

The **BOX** option allows you to use either the crossing or window rectangle to select objects. Define the rectangle by picking its opposite corners.

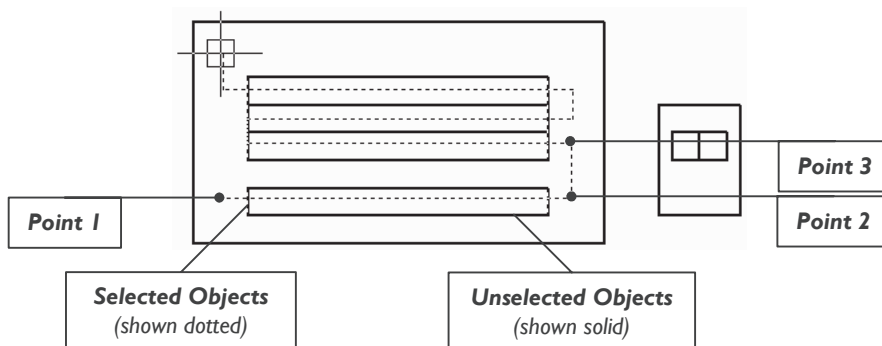
After you pick the first corner of the box, move the cursor either to the right or to the left. If you move to the right, the result is the **Window** selection: AutoCAD selects objects completely within the blue selection rectangle.

When you move to the left, the result is the crossing selection: AutoCAD selects objects within the rectangle, as well as those crossing it. In addition, the rectangle is drawn with dashed lines and filled with green color.



F (Select with a Fence)

The **F** option (short for “fence”) uses a polyline to select objects. The fence is displayed as a dashed line; all objects crossing the fence are selected. You can construct as many fence segments as you wish, and can use the **Undo** option to undo a fence line segment.



Select objects: **f**

First fence point: (*Pick point 1.*)

Specify endpoint of line or [Undo]: (Pick point 2.)

Specify endpoint of line or [Undo]: (Pick point 3.)

et cetera

AU (Select with the Automatic Option)

The **AU** option (short for “automatic”) combines the pick and **Box** options. It is the default for most selection operations. After you enter **AU** in response to the “Select objects” prompt, you select a point with the pickbox. If an object is found, the selection is made.

If an object is not found, the selection point becomes the first corner of the **Box** option. Move the box to the right for Window, or to the left for Crossing. The **AU** option is excellent for advanced users who wish to reduce the number of modifier selections.

WP (Select with a Windowed Polygon)

The **WP** option (short for “windowed polygon”) selects objects by placing a polygon window around them. The polygon window selects in the same manner as the **Window** option: all objects completely within the polygon are selected. Objects that cross or are outside the polygon are not selected.

The difference is that the **WP** option creates a multisided window, instead of a rectangle. Let's look at a sample command sequence:

Select objects: **wp**

First polygon point: (*Pick point 1.*)

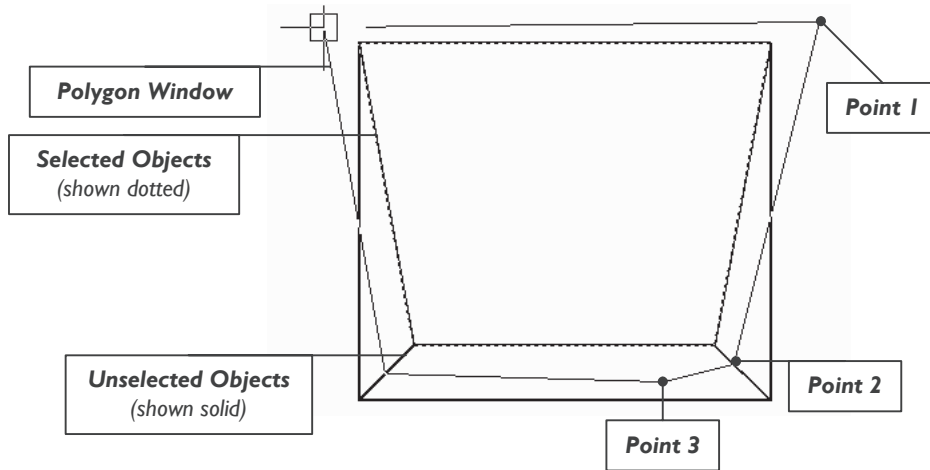
Specify endpoint of line or [Undo]: (Pick point 2.)

Specify endpoint of line or [Undo]: (Pick point 3.)

After you enter the first polygon point, build the window by placing one or more endpoints. The polygon rubber-bands to the cursor intersection, always creating a closed polygon window filled with the color blue.

Undo the last point entered by typing **U** in response to the prompt. Pressing **ENTER** closes the polygon window and completes the process.

Note that the polygon window must not cross itself or rest directly on a polygon object. If it does, AutoCAD warns “Invalid point, polygon segments cannot intersect,” and refuses to place the vertex.



CP (Select with a Crossing Polygon)

The **CP** option (short for “crossing polygon”) works in the same manner as the **WP** option, except that the polygon functions in the same manner as a crossing window. All objects within or crossing the polygon are selected. The crossing polygon is displayed with dashed lines filled with green, similar to a crossing window.

SI (Select a Single Object)

The **SI** option (short for “single”) forces AutoCAD to issue a single “Select objects:” prompt. (All other selection modes repeat the “Select objects:” prompt until you press **ENTER**.) You can use other selection options in conjunction with **SI** mode, such as the Crossing selection shown in this example:

```
Select objects: si
Select objects: c
Specify first corner: (Pick point 1.)
Specify opposite corner: (Pick point 2.)
nnn found
```

The Single option is useful for efficient single object selection in macros, because it does not require pressing **ENTER** to end the object selection process.

SU and O (Select Subobjects and Objects)

The **SU** option (short for “subobjects”) selects faces, edges, and vertices of 3D models. This is equivalent to holding down the **CTRL** key during object selection. The **O** option (short for “objects”) exits subobject selection mode.

M (Select Through the Multiple Option)

Each time you select an object, AutoCAD scans the entire drawing to find the object. If you are selecting an object in a drawing that contains a large number of objects, there can be a noticeable delay.

The **M** option (short for “multiple”) forces AutoCAD to scan the drawing just once. This results in shorter selection times. Press **ENTER** to finish the object selection and begin the scan. Note that AutoCAD does not highlight the objects selected until you press **ENTER**.

```
Select objects: m
Select objects: (Pick three times.)
3 selected, 3 found
```

P (Select the Previous Selection Set)

The **P** option (short for “previous”) uses the previous selection set. This very useful option allows you to perform several editing commands on the same set, without reselecting them. In addition, you can add to and remove from the **Previous** selection set.

L (Select the Last Object)

The **L** option (short for “last”) selects the last object drawn still visible on the screen. When the command is repeated, and **Last** is used a second time, AutoCAD chooses the same last object and reports “1 found (1 duplicate), 1 total.” This option is useful for immediately editing an object just drawn.

G (Select the Group)

The **G** option (short for “group”) adds the members of a named group to the selection set. (Groups are covered later in this chapter.) The option prompts:

Enter group name: *(Enter a group name.)*

Changing Selected Items

You can add and remove objects from the group of selected objects by using modifiers. Modifiers must be entered after you select at least one object, and before you press **ENTER** to end object selection.

U (Undo the Selected Option)

The **U** option (short for “undo”) removes the most recent addition to the set of selections. If the undo is repeated, you will step back through the selection set. This shortcut replaces the two-step process of using the **Remove** option, and then remembering the objects to pick.

R (Remove Objects from the Selection Set)

The **R** option (short for “remove”) removes objects from the selection set by any object selection method.

Select objects: **r**

Remove objects: *(Pick an object.)*

1 found, 1 removed, 2 total



Notes The **Remove** option is useful when you need to select a large number of objects with the exception of one or two objects located in the area. Select all the objects, then use the **R** option to remove the excess objects.

SHIFT+Select removes objects in **Add** mode, but adds objects in **Remove** mode.

A (Add Objects to the Selection Set)

The **A** option (short for “add”) adds objects to the set. **Add** is usually used after **Remove**. **Add** changes the prompt back to ‘Select objects:’ so you may add objects to the selection set. Hold down the **SHIFT** key to remove objects.

Canceling the Selection Process.

Pressing **ESC** at any time cancels the selection process and removes the selected objects from the selection set. The prompt line returns to the ‘Command:’ prompt.

Pressing **ENTER** ends the selection process, and continues with the editing command’s other options.

Selecting Objects During Commands

To use the selection options at any ‘Select objects:’ prompt, enter one of the abbreviations listed in the table on the earlier page (abbreviations shown in uppercase letters). Alternatively, just pick two points, because the **Auto** mode is the default.

For example, to move all the objects within the selection rectangle, do the following

Command: **move**

Select objects: **w**

Specify first corner: *(Pick point 1.)*

Specify opposite corner: *(Pick point 2.)*

5 found Select objects: *(Press ENTER to end object selection.)*

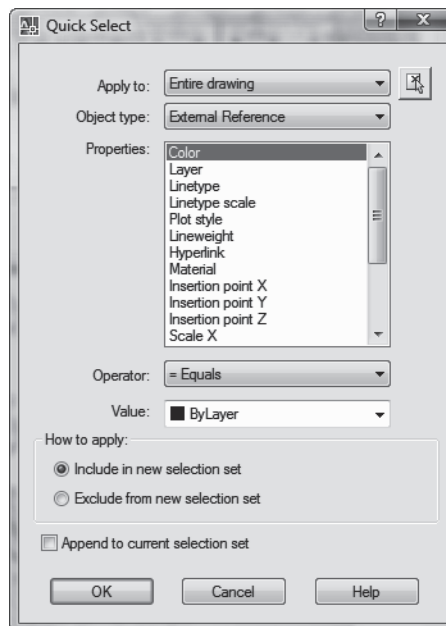
Specify base point or displacement: *(Pick a point.)*

Specify second point of displacement or <use first point as displacement>: *(Pick another point.)*

Notice that the “Select objects:” prompt repeats until you press **ENTER**. This means you can keep selecting (and deselecting) objects until you are satisfied with the selection set.

QSELECT

The **QSELECT** command selects objects based on their properties, instead of on their location, as with the **SELECT** command. For example, use the command to select all objects with hidden linetypes on a specific layer, or all circles with a radius larger than one inch.



The dialog box is also accessible through the **Quick Select** button in several other commands, such as **BLOCK** and **PROPERTIES**.

TUTORIAL: SELECTING OBJECTS BY PROPERTIES

- I. To select objects by their common properties, start the **QSELECT** command with one of these methods:
 - In the ribbon's 2D Home tab, choose the **Quick Select** button from the Utilities panel.
 - Right-click, and from the shortcut menu choose **Quick Select**.

- At the ‘Command:’ prompt, enter the **qselect** command:

Command: **qselect** (Press ENTER.)

Notice the Quick Select dialog box.

2. The selection can be made from the entire drawing or a subset of the drawing, called the “current selection.”

To select a subset, click the **Select Objects** button next to the **Apply to** droplist.

AutoCAD prompts you:

Select objects: (Select one or more objects.)

Select objects: (Press ENTER to return to the dialog box.)

Notice that the Apply To droplist now has two options. Select one:

- Entire Drawing.
 - Current Selection.
3. The Object Type droplist allows you to narrow down the selection to specific objects. Only the objects found in the drawing or the current selection are listed here. To include all objects, select “Multiple.”
 4. Narrow down your selection by picking one of the items in the Properties list. The object you selected earlier affects the content of this list, which contains only properties specific to the object. You can select only one property from this list.
 5. The Operator list further narrows the range AutoCAD searches for. Depending on the property you select, the choice of operator includes:

Operator	Meaning
=	Equals
<>	Not Equal To
>	Greater Than
<	Less Than
*	Wildcard Match (selects all).

Use the *Wildcard Match operator with text fields that can be edited, such as the names of blocks.

6. The Value field goes with the Operator list: if AutoCAD can determine values, it lists them here; if not, you type the value.
7. Under How to Apply, include or exclude objects matching these parameters from the existing selection set.
8. The Append to Current Selection Set check box determines whether these are added to (on) or replace (off) the current selection set.
9. Choose **OK** to exit the dialog box. AutoCAD highlights the objects selected, and reports the number selected:
nnn item(s) selected.

CONTROLLING THE SELECTION: ADDITIONAL METHODS

In addition to selection modes and direct selection, AutoCAD has these commands:

- **PICKBOX** changes the size of the square pick cursor.
- **DDSELECT** changes selection options; displays the Selection tab of the Options dialog box.
- **FILTER** selects objects based on their properties.
- **SELECTURL** highlights all objects that contain hyperlinks.

Let's look at how these commands work.

Pickbox

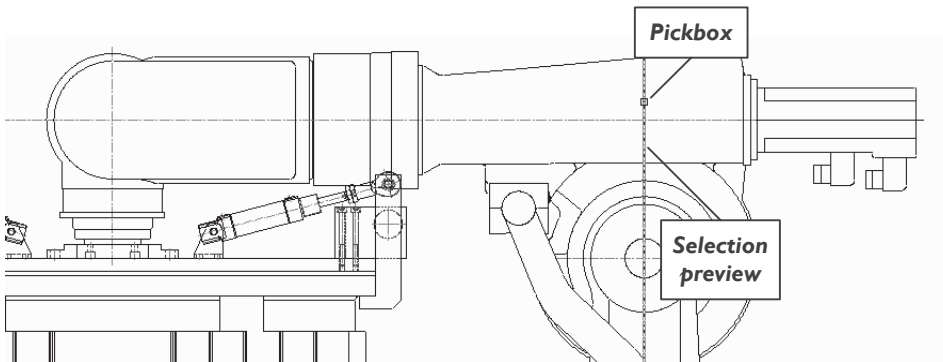
When an editing command displays the “Select objects:” prompt, the cursor is supplemented by a small square called the “pickbox.” When you place the pickbox over the object, and then click (press the left mouse button), AutoCAD scans the drawing, and selects the object under by the pickbox.

The pickbox can be made larger and smaller. The Selection tab of the Options dialog box allows you to change the pickbox size; or, you just might prefer to use the PICKBOX system variable:

Command: **pickbox**

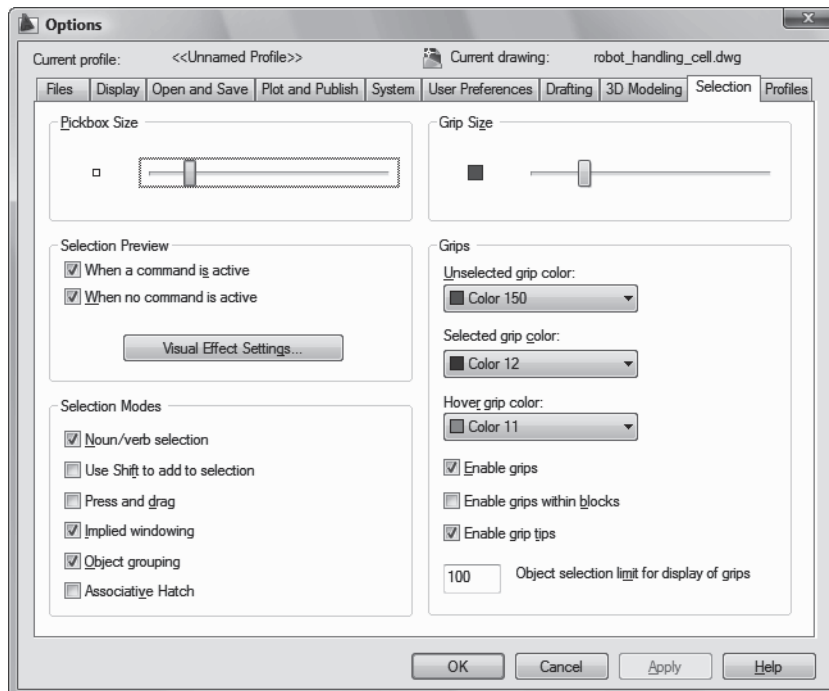
Enter new value for PICKBOX <3>: *(Enter a value between 0 and 50.)*

The size of the pickbox is measured in pixels. A large pickbox forces AutoCAD to scan through more objects, which can take longer in complex drawings on slow computers.



DdSelect

The DDSELECT command allows you to change selection modes. (The command is actually an alias for displaying the **Selection** tab of the Options dialog box.)



- Right-click, and from the shortcut menu choose **Options**.
- At the ‘Command:’ prompt, enter the **ddselect** command.

Command: **ddselect** (Press ENTER.)

Let's look at each option.

Pickbox Size

The pickbox is the square cursor that appears at the 'Select objects:' prompt. Here, you can change its size from small to large.

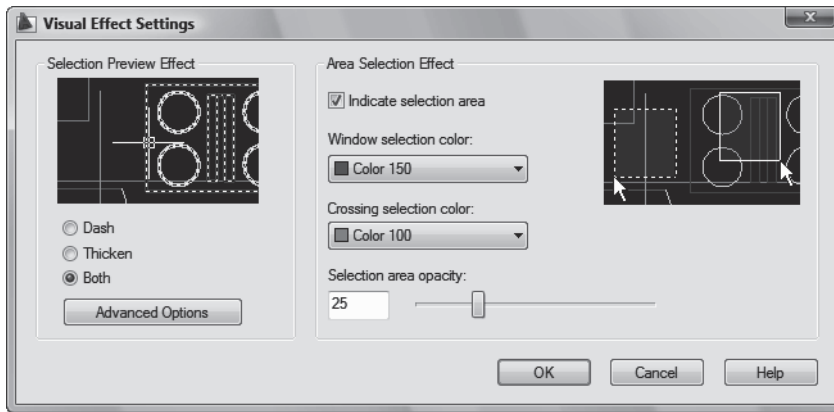
Selection Preview

Selection preview highlights objects as the pickbox cursor moves over them. You can turn the mode on and off.

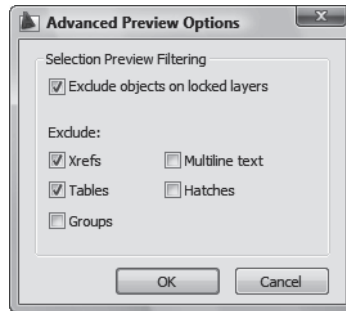
When a Command Is Active means that selection preview works only when a command displays the 'Select objects:' prompt.

When No Command Is Active means that selection preview works any other time.

The **Visual Effect Settings** button displays a dialog box that controls the visual effects of selection previews and windowed selections. For selection previews, you can choose whether dashed lines, thickened lines, or both are displayed. For windowed and crossing selections, you can choose the color and opacity. (Autodesk documentation is incorrect in suggesting these affect the appearance of windowed areas only during selection preview.)



The **Advanced Options** button accesses further options, specifically which objects should be excluded from selection previewing.



Noun/Verb Selection

The "normal" way to edit drawings is to (1) first select the editing *command*, and then (2) select the *objects* to modify. Technically, this is choosing the *verb* (the action represented by the edit command), and then the *noun* (the object of the action represented by the selection set).

As an alternative, AutoCAD allows you to reverse the procedure: (1) first select the *object(s)* to edit, and then (2) choose the editing *command*. This is called "noun/verb selection."

When the **Noun/Verb Selection** option is turned on (as it is by default), AutoCAD places the pickbox at the intersection of the crosshairs. The presence of the pickbox indicates that you can select objects before entering one of these editing commands:

ALIGN LIST ARRAY MIRROR BLOCK MOVE CHANGE PROPERTIES CHPROP
ROTATE COPY SCALE DVIEW STRETCH ERASE WBLOCK EXPLODE BEDIT

Here is the command sequence to erase a single object, for example:

Command: *(Select the object.)*

Command: **erase**

1 found

Use Shift to Add to Selection

When selecting, you normally choose the object(s), and then press ENTER when you are finished. As you select each object, it is automatically added to the selection set. To remove objects from the selection set, hold down the SHIFT key while picking them.

As an alternative, turn on the **Use Shift to Add to Selection** option. Now you must hold down the SHIFT key to add objects to the selection set; this is similar to the method used by many other Windows programs. When you make more than one selection without holding the SHIFT key, the previous selections are removed from the selection set.

Press and Drag

The traditional AutoCAD method for selecting objects through windowing is to (1) click one corner, (2) move the cursor to the other corner, and then (3) click the other corner.

As an alternative, turn on the **Press and Drag** option. You build a window by (1) clicking one corner, and then (2) dragging (by holding down the mouse button) the cursor to the other corner. Like the **Shift to Add** option, this is used by most other Windows programs.

Implied Windowing

You previously learned how to use a selection window or a crossing window to select objects. All you have to do is enter either a W or a C to invoke the window mode.

The **Implied Windowing** option does this automatically. (The option is on by default.) You “imply” the window by clicking in an empty area of the drawing. When AutoCAD does not find an object within the area of the pickbox, it assumes that you want to use windowing. When you move the cursor to the right, Window selection mode is entered; move to the left, Crossing selection mode is entered.

Because the first point entered describes the first corner of the window, be sure to select a desirable position.

Object Grouping

When the **Object Grouping** option is turned on, AutoCAD selects the entire group when you pick one object in the group. When off, just the picked object is selected. (More on groups follows in this chapter.)

As a shortcut, you can toggle object grouping mode with CTRL+SHIFT+A. Each time you press the key combination, AutoCAD comments:

Command: *(Press CTRL+SHIFT+A.)* <Group off>

“Group off” means objects are selected, while “Group on” means the entire group is selected:

Command: *(Press CTRL+SHIFT+A.)* <Group on>

Curiously, CTRL+H does the same thing. Technically, it toggles the PICKSTYLE system variable between

0 (individual objects selected from the group) and 1 (entire group selected). PICKSTYLE can also have the values of 2 and 3 for toggling the selection of associative hatch patterns.

Associative Hatch

When the **Associative Hatch** option is turned on, AutoCAD selects the boundary object(s) when you pick an associative hatch pattern. When off, only the hatch is selected.

Filter

The **FILTER** command displays a dialog box that selects objects based on their properties. You can save the resulting selection set by name, and then access it at the 'Select objects:' prompt.

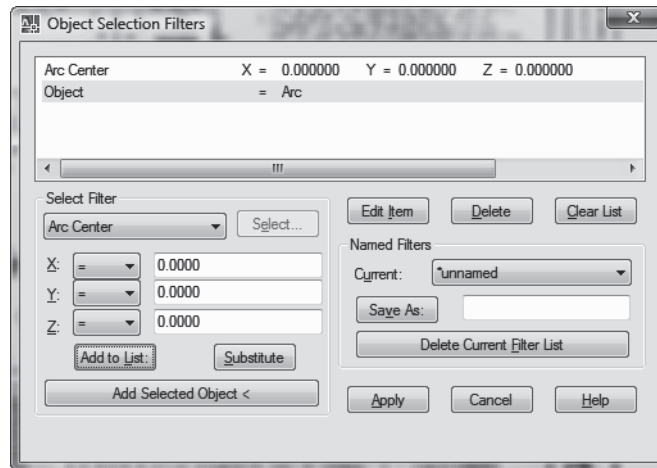
- At the 'Command:' prompt, enter the **filter** command.
- Alternatively, enter the **f** alias at the keyboard.

Command: **filter** (Press ENTER.)

- Because **FILTER** is a *transparent* command, it can be invoked during another command to filter the selection set:

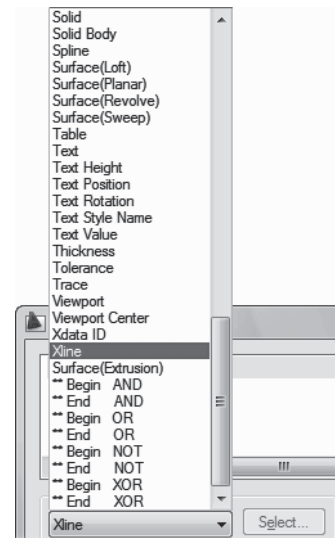
Command: **erase**

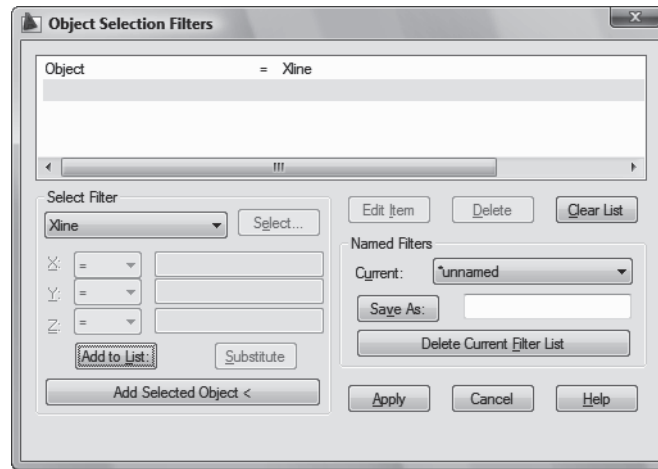
Select objects: '**filter**



The selection set created by **FILTER** can be accessed via the **P** (previous) selection option. In the following tutorial, you erase all construction lines from a drawing:

1. Start the **ERASE** command:
Command: **erase**
2. Then invoke the **FILTER** command transparently by prefixing the command with the quotation mark:
Select objects: '**filter**
Notice that AutoCAD displays the Object Selection Filters dialog box.
3. In the Select Filter droplist, select **Xline**.
Xline is near the end of the list. Here's a quick way to get to any item in an alphabetical list: after clicking on the droplist, press **x** on the keyboard. You are taken to the first word starting with "x".
4. Click the **Add to List** button. Notice that AutoCAD adds this text to the "list" (the large white area at the top of the dialog box):
Object =Xline





5. Click **Apply**. Notice that the dialog box disappears, and that the **FILTER** command displays prompts on the command line:

Applying filter to selection.

This means that AutoCAD will apply the filter (search for all xlines) to the objects you now select.

6. Specify that the filter should apply to the entire drawing with the **All** option:
 >>Select objects: **all**

9405 found 9401 were filtered out.

The double angle bracket (>>) indicates AutoCAD is currently in a transparent command. (The **ERASE** command continues later.)

7. Return to the **ERASE** command by exiting the **FILTER** command. Press **ENTER**:
 >>Select objects: (Press **ENTER** to exit the **FILTER** command.)

Exiting filtered selection.

Resuming **ERASE** command.

AutoCAD returns to the **ERASE** command, and picks up the four objects selected by the **FILTER** command:

Select objects: 4 found

8. Press **ENTER** to erase the four objects and exit the **ERASE** command:
 Select objects: (Press **ENTER** to exit the **ERASE** command.)

AutoCAD uses the selection set created by the **FILTER** command to erase the xlines.

In addition to filtering objects, this command also filters x, y, z coordinates, such as the center point of circles. Other options become available in the dialog box as you select them. For example, when you select **Elevation**, the X text entry box allows you to specify the elevation using the following operators:

Operator	Meaning
<	Less than.
<=	Less than or equal to.
=	Equal to.
!=	Not equal to.
>	Greater than.
>=	Greater than or equal to.
*	All values.

Specifying elevation > 100, for example, means that all objects with an elevation greater than 100 will be added to the filtered selection set.

In addition, you can group filter sequences using these operators:

Operator	Meaning
**Begin OR with **End OR	Include <i>any</i> of these items.
**Begin AND with **End AND	Include <i>all</i> of these items.
**Begin NOT with **End NOT	Include <i>none</i> of these items.
**Begin XOR with **End XOR	Include none of these items if one item is found.

You can save filter definitions by name to .nfl (short for “named filter”) files on disk for use in other drawings or editing sessions.

 DRAWORDER

The **DRAWORDER** command determines the order in which overlapping objects are displayed. In the figure below, the text “Using AutoCAD” is obscured by the rectangle. By changing the display order — the order in which AutoCAD redraws objects — you can make the text visible on top of the inappropriately placed rectangle.



Left: Text above the rectangle.
Right: Text below the rectangle.

TUTORIAL: SPECIFYING DRAWORDER

- To specify the display order of overlapping objects, start the **DRAWORDER** command with one of these methods:
 - In the ribbon’s 2D Home tab, choose the **Bring to Front** button from the Modify panel.
 - At the ‘Command:’ prompt, enter the **draworder** command.
Command: **draworder** (Press ENTER.)
 - Alternatively, enter the **dr** alias at the keyboard.
- Select one or more objects:
Select objects: (Select one or more objects.)
1 found Select objects: (Press ENTER to end object selection.)
- Specify the display order for the selected object(s)
Enter object ordering option [Above object/Under object/Front/Back] <Back>: (Press ENTER.)
Regenerating model.

CHANGING THE DRAW ORDER: ADDITIONAL METHODS

The **DRAWORDER** command has four options for controlling the visual overlapping of objects. The first two listed below are useful when three or more objects overlap.

- Above object** places the object visually on top of other selected objects.
- Under object** places the object under other selected objects.

- **Front** places the object above all other objects.
- **Back** places the object below all other objects.

HpDrawOrder

AutoCAD provides additional control over draw order. The **HPDRAWORDER** system variable determines whether hatch and fill patterns are above or below their boundaries and other objects. See the **HATCH** command earlier in this chapter.

TextToFront

The **TEXTTOFRONT** command determines whether text and/or dimensions are always placed on top (or in front) of overlapping objects in the drawing.

Command: **texttofront**

Bring to front [Text/Dimensions/Both] <Both>: (Press ENTER.)

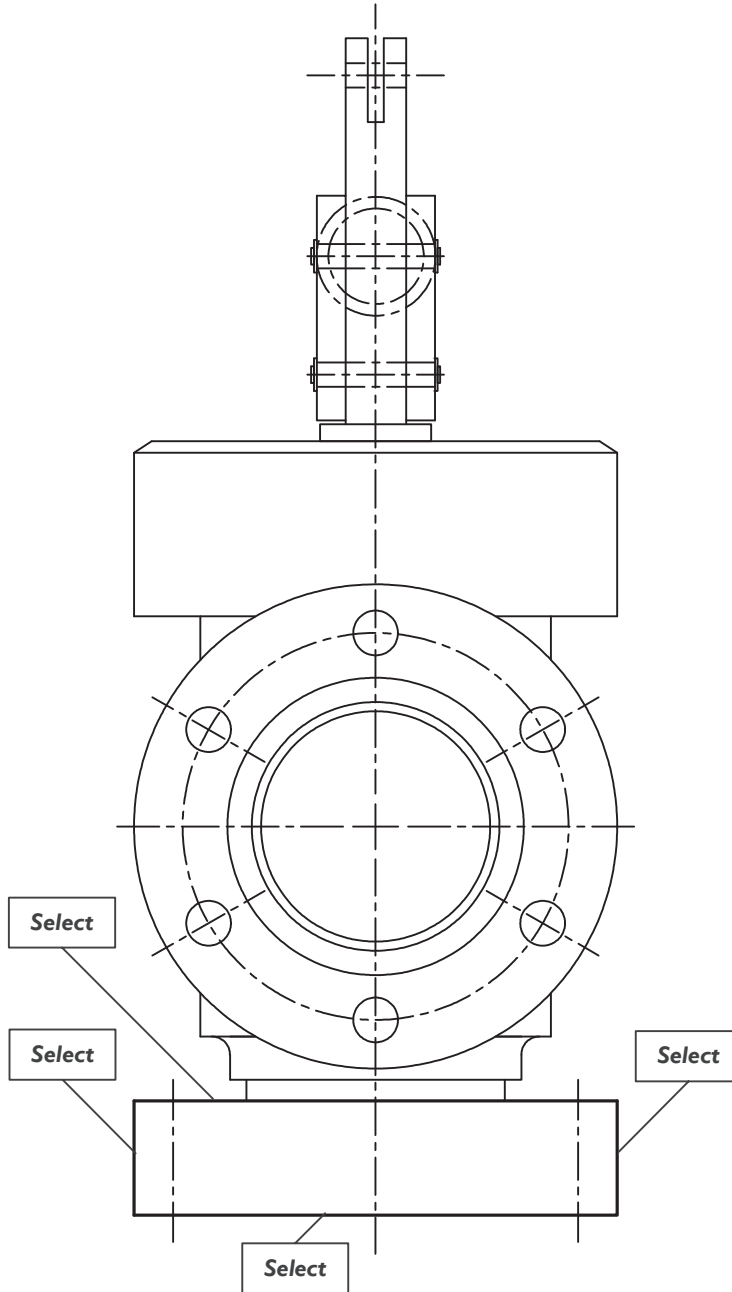
This command ensures the text is always legible, not obscured by other objects. It applies to all text and dimensions in the drawing. It does not, however, work when model space objects overlay text in a layout, and vice versa.

This command is not a mode; if text and dimensions are obscured by later objects, you must run this command again.

EXERCISES

- I. In this exercise, you use some of the options of the **SELECT** command to erase portions of a valve housing drawing.

From the Companion CD, open the *edit1.dwg* drawing.



Use the **ERASE** command to delete some of the objects.

You should now see a pickbox on the screen. Place the pickbox over the bottom line of the part, as shown in the illustration, and click. The line should be highlighted.

One by one, select the remaining lines, as shown in the figure.

Finally, press **ENTER**. The four lines are removed from the drawing.

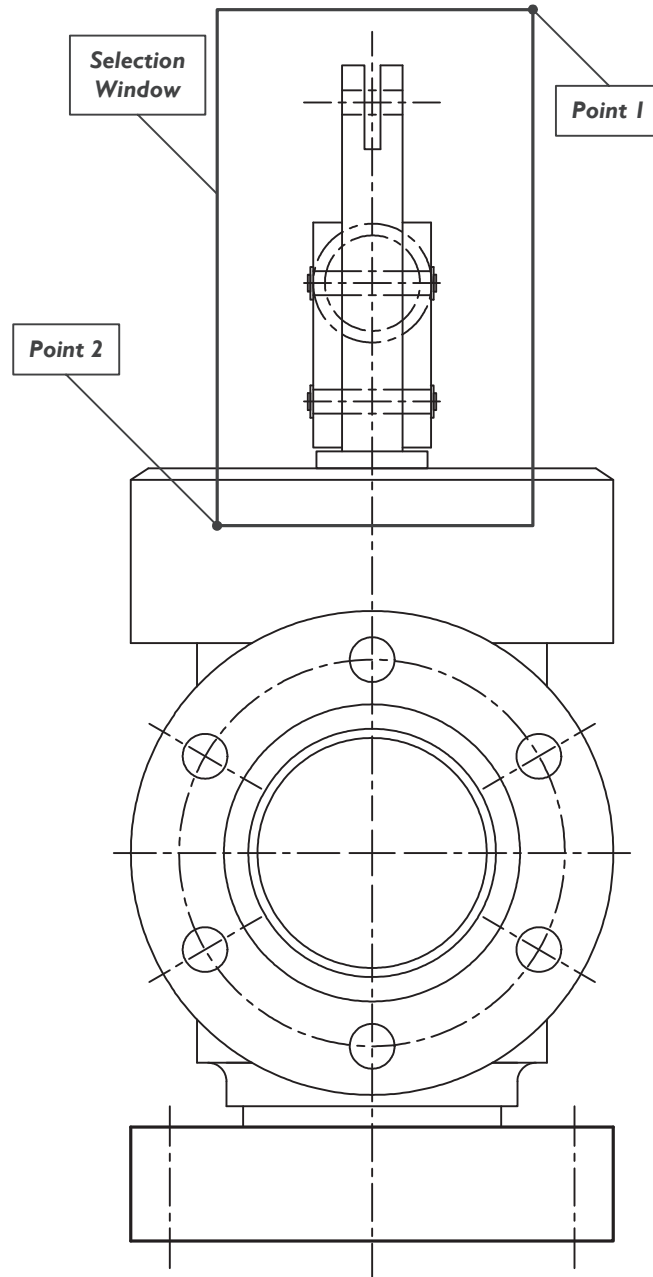
2. Repeat the above exercise on the same drawing with the **Window** object selection mode. But first, use the **u** command to undo the erasure. Select **ERASE** again, and this time enter **w** in response to the “Select objects:” prompt. Consult the figure for the points referenced in the following command sequence.

Command: **erase**

Select objects: **w**

Specify first corner: *(Select point 1.)*

Specify other corner: *(Select point 2.)*



All items within the window are selected and highlighted. Notice that objects that extend outside the window area (but are not wholly contained therein) are not selected.

Select objects: *(Press ENTER.)*

Press **ENTER** and the selected objects are deleted.

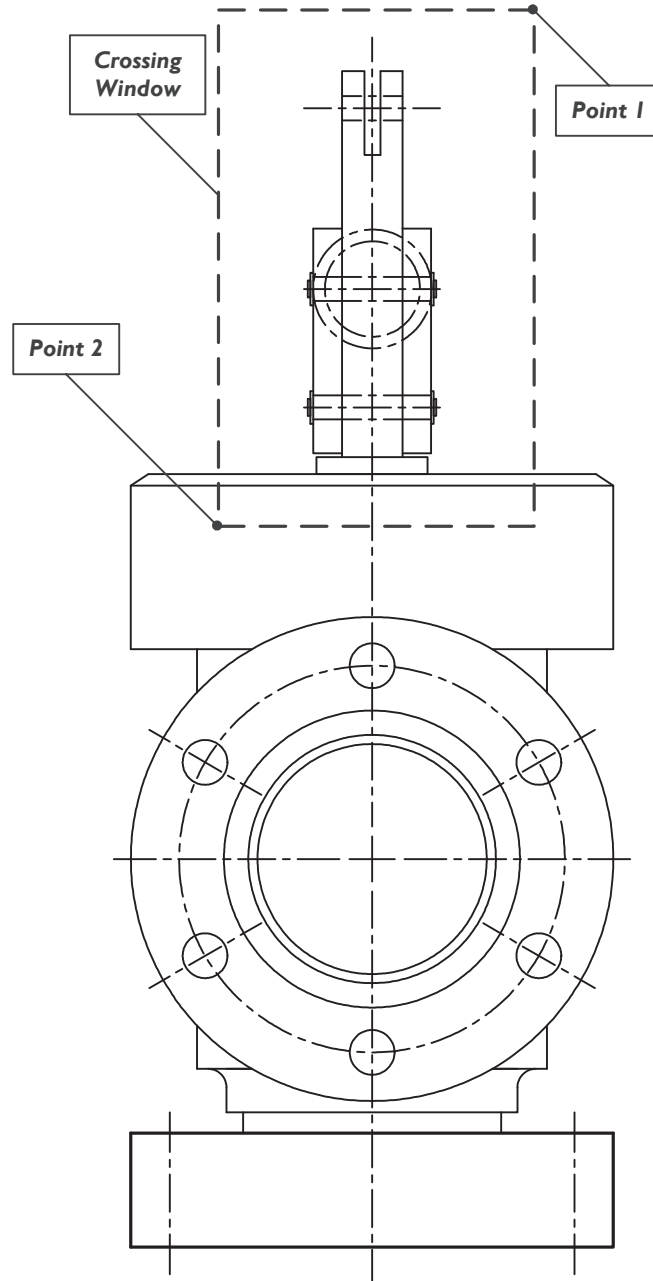
3. Repeat the above exercise on the same drawing with the **Crossing** window object selection mode. Again, first undo the erasure with the **u** command. Select **ERASE** again, entering **C** (for “crossing”) as the option. Refer to the following command sequence and the figure.

Command: **erase**

Select objects: **c**

Select first corner: *(Select point 1.)*

Select other corner: *(Select point 2.)*



Notice that all the objects within and crossing the window are selected (as noted by the highlighting).

4. Continuing from the above exercise, remove objects from the selection set.
After you placed the crossing window in the previous exercise, AutoCAD asked you to select more objects. This time, enter **R** for remove. The command sequence continues:

Select objects: **r**

Remove objects: *(Select one of the horizontal lines.)*

1 found, 1 removed Remove objects: *(Select the other horizontal line.)*

1 found, 1 removed Remove objects: *(Press ENTER.)*

The objects you removed from the object selection set are not erased.

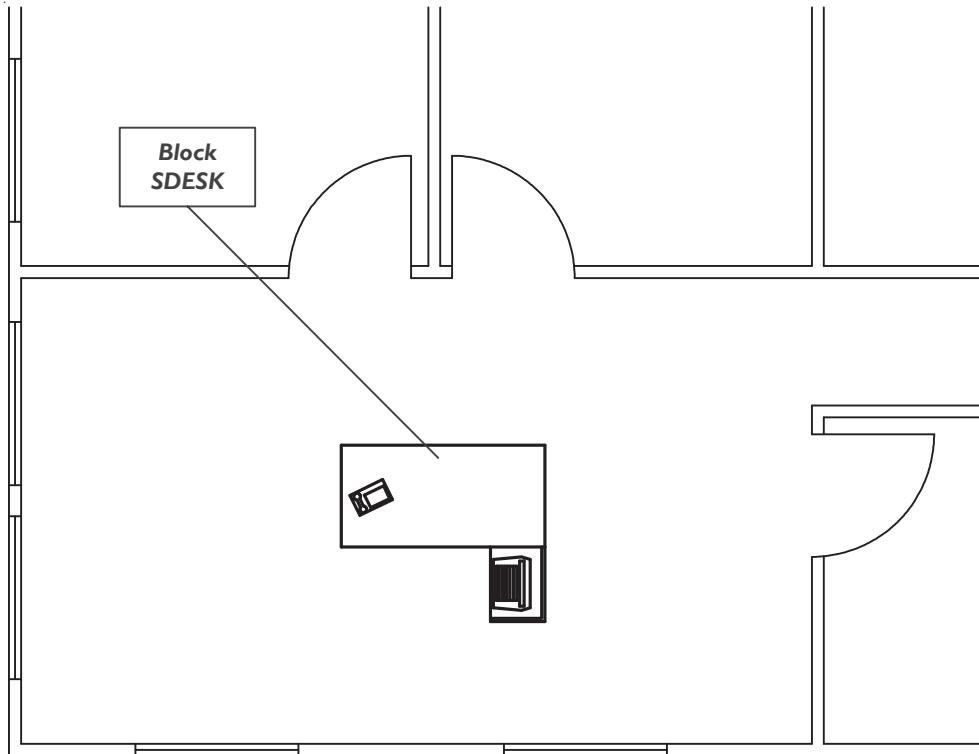
Exit the drawing, discarding the changes you made so that the edits are not recorded.

5. In this exercise, you create a block from objects within a drawing, and then insert the block in the drawing several times.

From the Companion CD, open the drawing named *office.dwg*.

Window the entire desk, and then start the **BLOCK** command.

Name the block "SDESK", and then pick an insertion point.



Click **OK**. The block is now stored with the drawing, and can be used as many times as you require.

With the **INSERT** command, place the SDESK block in the drawing three times, using these scales and rotation angles:

Scale = 1.0	Angle = 0 degrees
Scale = 1.5	Angle = 90 degrees
Scale = 0.75	Angle = 45 degrees

6. In this exercise, use DesignCenter to place blocks in a drawing.
From the Companion CD, open the site plan drawing named *insert.dwg*. The landscape items are drawn for you and stored as blocks in the drawing.
Use **INSERT** to view blocks, and then insert them in the drawing to create a landscaping design of your own.
7. In this exercise, you draw an electronic part called a “diode,” and then insert it as an array of blocks.
Using the **PLINE** command, construct the semi-conductor symbol shown below.

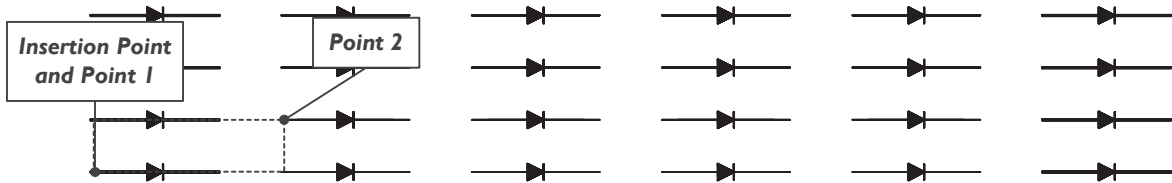


With the **BLOCK** command, select the diode, and name it “DIODE”; use the left end of the diode as the insertion point.

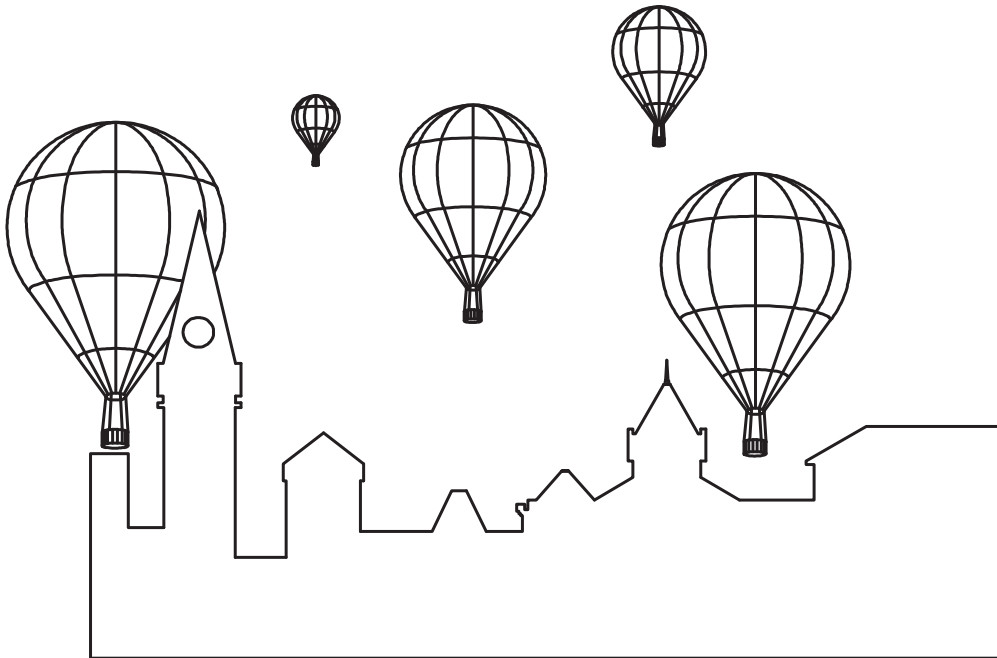
You have now created the block DIODE in the drawing.

Use the **INSERT** command, followed by the **ARRAY** command to place a 4x6 array of diodes.

You may need to use the **Zoom All** command to see the completed array.

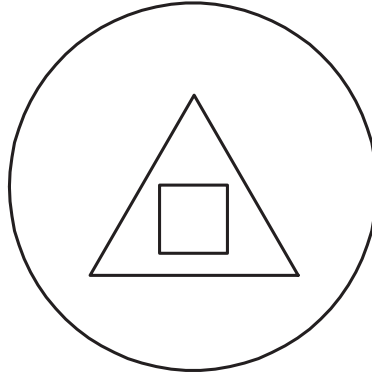


8. In this exercise, you hatch portions of a drawing of several hot air balloons.
From the Companion CD, open the drawing named *solids.dwg*.
Use the **HATCH** command to place the solid hatch pattern in a variety of colors in some of the balloon areas.



Now try placing a variety of hatch patterns in the same drawing.

9. In this exercise, you specify the area to be hatched with a pick point. The figure shows an object containing three areas that could be hatched.



You need to hatch the square and circle, but not the triangle.

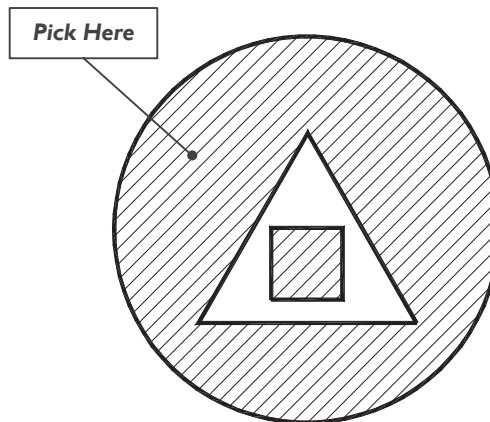
Draw the circle, triangle, and square using the **CIRCLE** and **POLYGON** commands.

From the **Draw** menu, select **Hatch**.

In the **Pattern** list box, select **ANSI31**.

Select the **Advanced** tab, and then ensure that **Normal** appears in the **Style** list box and that the **Island Detection** method has **Flood** selected.

Select the **Pick Points<** button, and then click inside the circle, but outside the triangle



Right-click to return to the dialog box.

Choose the **Preview** button. Your hatch should look similar to the figure above. If not, change the hatch options and/or your pick points.

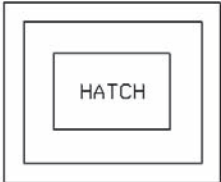
When the preview hatch looks right, choose **OK** to apply the hatch pattern.

10. In this exercise, you use hatch styles to control the behavior of hatch patterns.

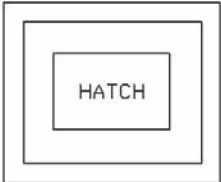
From the Companion CD, open the *hatch.dwg* drawing file.

Start the **HATCH** command, and set the following options:

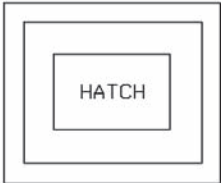
- Pattern **ANSI31**
- Scale **1**
- Angle **0**
- Island Detection Style **Normal**
- Select objects **W** (or pick a point between the two outer rectangles)



NORMAL

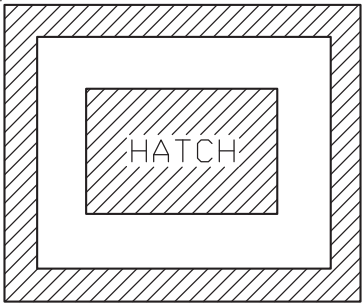


OUTERMOST



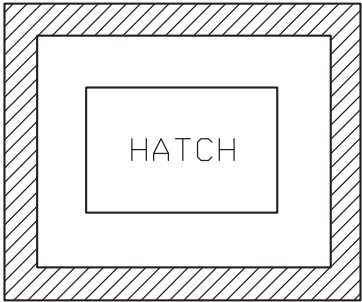
IGNORE

Place the window around the object, and then press **ENTER**. Your drawing should look similar to the following illustration:



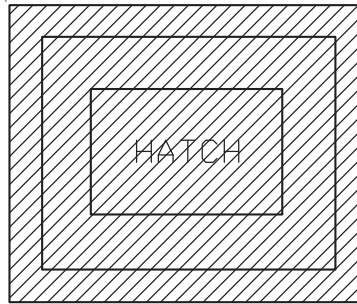
NORMAL

11. Repeat the above exercise on the same drawing, but use the **Outer** style. The drawing should look similar to the following illustration:



OUTERMOST

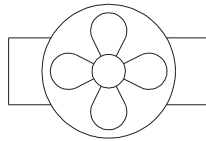
12. Repeat the above exercise on the same drawing, but use the **Ignore** style of island detection. Notice that the hatch has ignored the boundaries and the text. Your drawing should look similar to the following illustration:



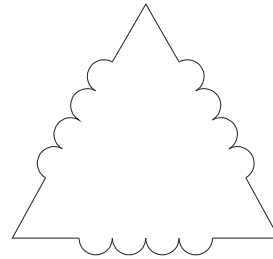
IGNORE

13. Use the Block Editor to design the following symbols. Save them using the name indicated.

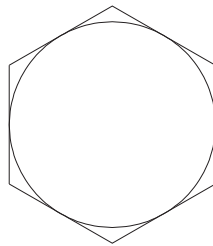
a. Fan



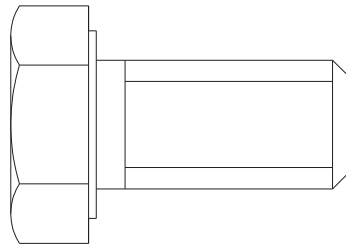
b. TransformerDelta



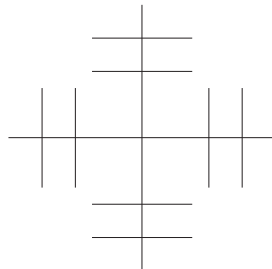
c. HexHeadScrew-Top



d. HexHeadScrew-Side



e. FlangedCross



f. ElbowOutlet-Up



CHAPTER REVIEW

1. Describe the purpose of DesignCenter.
2. What are two ways to insert a block from DesignCenter into a drawing?
 - a.
 - b.
3. Describe two ways to change blocks to a new definition.
4. Which key do you hold down to select more than one item?
5. Explain the effect of turning off the **OSNAPHATCH** system variable.
6. Can hatch patterns be trimmed?
Can hatch patterns be used as trim boundaries?
7. How would you force text and dimensions to display in front of other overlapping objects in drawings?
8. Name three uses for the Block Editor:
 - a.
 - b.
 - c.
9. Explain what the **HATCH** command does.
10. If you did not want the solid filled areas in your drawing to plot, what could you do?
11. What are *hatch boundaries*?
12. What are *blocks*?
13. What is the *base point* of a block?
14. When placed in drawings, how do blocks handle their layer definitions?
15. What is a *nested block*?
16. How would you create a separate drawing file from an existing block?
17. How do you place blocks in drawings?
18. How can you place blocks in other drawings?
19. When you place blocks in the drawings, what is the *insertion point*?
20. Can you place one AutoCAD drawing in another AutoCAD drawing?
21. Name two advantages of using blocks.
22. What commands are combined to create the **MINsert** command?
23. How would you change the origin point of a hatch?
24. When do hatch patterns obscure text?
25. When an editing command is invoked, must a selection option, such as **Single** or **Window**, be entered before an object is selected?
26. When choosing a set of objects to edit, other than by using the **U** command, how can you alter an incorrect selection without starting over?
27. If you wish to edit an item that was not drawn last, but was the last item selected, would the **Last** selection option allow you to select the item?
28. How do you increase and decrease the size of the pickbox?
29. When entering **BOX** in response to the “Select objects:” prompt, how are you then allowed to choose objects?
30. What makes it evident that an item has been selected?
31. When a group of items is selected with the **Window** option, does an item become a part of the selection set as long as it is partially inside the window?
32. In the object selection process, how is the **BOX** option different from the **Automatic** option?
33. What is the purpose of the tool palette?
34. How do you add blocks from a tool palette to the drawing?

35. Describe the purpose of the **BASE** command.
36. When might you use the Hatch and Gradient dialog box's **Draw Order** option?
37. What kind of hatches can the **HATCHEDIT** command not change?
38. Can the **HATCHEDIT** command copy the properties of one hatch pattern and apply them to another?
39. What is the difference between the **SELECT** and **QSELECT** commands?
40. Can you change the size of the *pick* cursor?
41. What is the purpose of the **HATCH** command's **Gap Tolerance** option?
42. What is the purpose of the **DRAWORDER** command?
43. Explain how *auxiliary scale* differs from the normal scale factor.

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Editing
Drawings

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CHAPTER 7

Properties and Layers

As you create drawings, you sometimes need to set the properties of objects, such as color, layer assignment, or linetype pattern. In this chapter, you learn how to modify the properties of any object using the following commands:

MATCHPROP matches the properties of one object to other objects.

COLOR changes the colors of objects.

LWEIGHT changes the display width of lines.

LINETYPE changes the patterns of lines.

LTSCALE and **CELTSIZE** change the scale factor of linetypes.

LAYER displays the Layer Properties Manager palette.

LAYERSTATE accesses the Layer States Manager dialog box.

SETBYLAYER resets layer properties in overridden viewports.

AI_MOLC and **CLAYER** set the current layer.

LAYERP and **LAYERPMODE** set the layers to their previous state.

LAYISO isolates selected layers by fading out all other layers.

PROPERTIES, **QUICKPROPERTIES**, and **ROLLOVERTIPS** report and/or change properties of selected objects.



MATCHPROP

The **MATCHPROP** command “reads” the properties of one object, and then applies them to other objects.

At times, you may want a group of objects to match the properties of another object. For example, you may accidentally place several doors on the Landscape layer, instead of the Door layer. This command lets you make this change. Or, you may realize that some lines drawn with hidden linetype should be in another linetype — whose name you don’t recall.

You could use the Properties toolbar to change layers, colors, linetype, and so on, but it doesn’t work well if you are not sure of the exact layer and linetype name. Complex drawings have hundreds of layer names, many of which look similar. One sample drawing provided with AutoCAD has the following layer names: ARCC, ARCCLR, ARCDIMR, ARCDSHR, ARCG, ARCM, ARCR, ARCRMNG, ARCRMNR, and ARCTXTG. And that’s just the first ten! Similarly, linetypes can look confusingly alike. (Layers, linetypes, and other properties are discussed later in this chapter.)

The **MATCHPROP** command solves those problems. It lets you copy the properties from one object to a selection set of objects. You must, however, be sure to select objects in the correct order: (1) select the single object whose properties to copy, and then (2) select the object(s) to take on those properties.

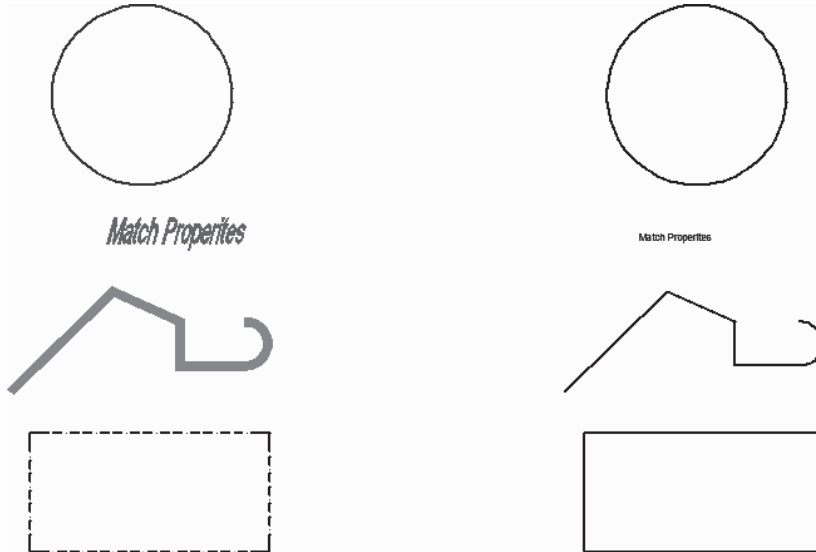
Property	Objects Affected
Color	All objects, except OLE objects.
Dimension	Dimensions, leaders, and tolerances; also paints dimension styles.
Hatch	Hatches; also paints the hatch pattern.
Layer	All objects, except OLE objects.
Linetype	All except attributes, hatches, multiline text, OLE objects, points, & viewports.
Linetype Scale	All objects except attributes, hatches, multiline text, OLE objects, points, and viewports.
Lineweight	All objects.
Material	All objects.
Multileader	Multileaders.
Plot Style	All OLE objects; unavailable when PSTYLEPOLICY is 1.
Polyline	Polylines; also paints width and linetype generation, but not fit/smooth, variable width, or elevation.
Shadow display	All objects.
Thickness	Arcs, attributes, circles, lines, points, 2D polylines, regions, text, and traces.
Text	Text and multiline text only; also paints the text style.
Viewport	Viewports; also paints on/off, display locking, standard or custom scale, shade plot, snap, grid, as well as UCS icon visibility and location settings, but not clipping, UCS-per-viewport, or layer freeze/thaw state.
Visual style	All objects.

TUTORIAL: MATCHING PROPERTIES

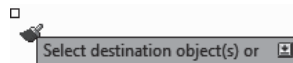
1. From the Companion CD, open the Match properties.dwg file, the drawing of several multi-colored objects — and equal number of monochrome objects.
2. To match the properties of one object to others, start the **MATCHPROP** command with one of these methods:
 - In the **Home** tab’s **Clipboard** panel, choose **Match Properties**.
 - At the ‘Command:’ prompt, enter the **matchprop** command:

Command: **matchprop** (Press ENTER.)

- Alternatively, enter the aliases **ma** or **painter** (the old name in AutoCAD LT) at the 'Command:' prompt.
- In all cases, AutoCAD prompts you at the command line:
Select source object: *(Pick a single object.)*



Notice that the cursor turns into a paintbrush.



On the command bar, AutoCAD lists the *active settings*, properties to be “picked up” by the paintbrush.

Current active settings: Color Layer Ltype Ltstyle Lineweight Thickness PlotStyle Dim Text Hatch Polyline Viewport Table Material Shadow display Multileader

- Select the objects to which the properties should be applied.
Select destination object(s) or [Settings]: *(Pick one or more objects, or enter S.)*
- You can continue selecting objects using windows, fences, single picks, and other selection modes.

When done, press **ENTER** to exit the command.

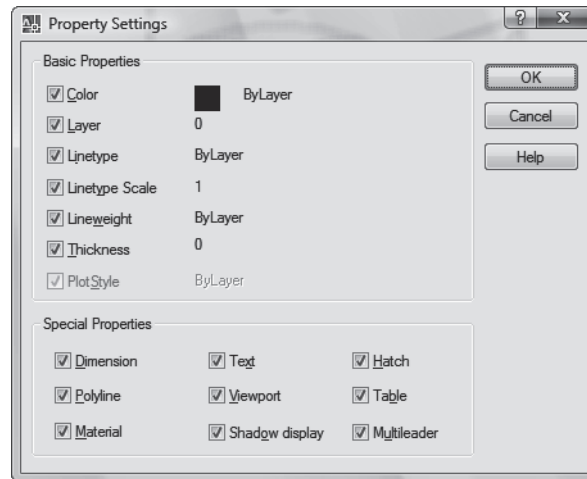
Select destination object(s) or [Settings]: *(Press ENTER.)*

Settings

At the “Select destination object(s) or [Settings]:” prompt, enter **S** to display the Property Settings dialog box.

The properties listed by this dialog box are more extensive than those listed by the Properties toolbar. Not all properties, however, work with all objects. For example, it makes no sense to match the hatch properties of text objects, since text cannot be hatched.

By default, all properties are on, meaning they will all be copied.



When you turn off properties, AutoCAD remembers this the next time you use **MATCHPROP**, which is why it lists “Current active settings.”

COLOR

Colors are used typically to identify objects or to segregate them visually.

In AutoCAD, the default color of objects is white or black, depending on the background color of the viewport. If the background is a dark color, then objects are displayed in white; if light, then AutoCAD automatically switches to black objects. (In drawings begun with the *acad3d.dwt* template file, the default color is pale blue.)

But, you don’t have to stick with black or white or pale blue. You can set colors before you draw, or change them afterwards:

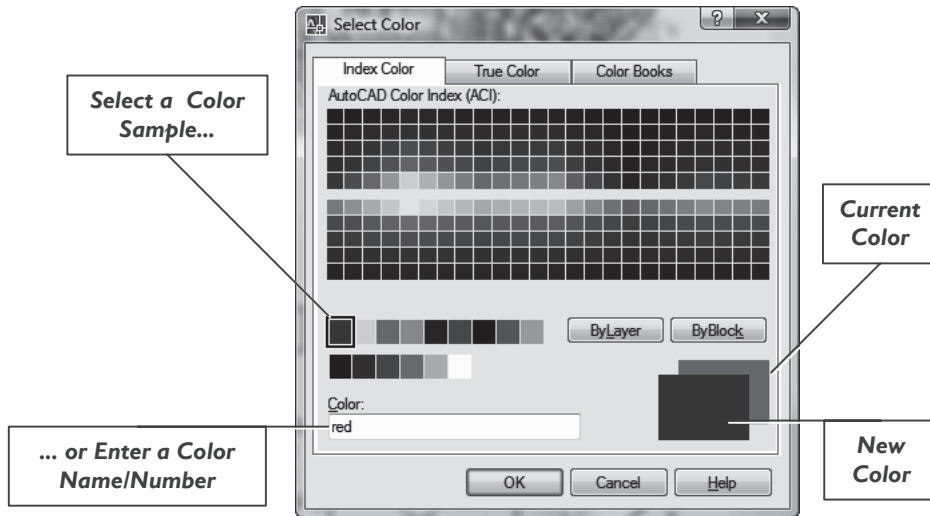
- **Presetting colors.** For objects you plan to draw, the best method is to specify colors through the Color option of the **LAYER** command; less preferred is to set the working color through the **COLOR** command.
- **Changing colors.** For objects already in the drawing, the *fastest* method is to use the Color Control droplist on the Object Properties panel; alternatives are to use the **PROPERTIES** or **CHPROP** command. The *best* method, however, is to move the objects to the appropriate layer, so that they take on the color of the layer.

It is important to understand that the **COLOR** command overrides the colors set by the **LAYER** command. Here is the difference between them: **LAYER** sets the color for objects drawn on each layer; **COLOR** sets the color for *subsequently* drawn objects. Thus, it is possible for a layer to contain objects of different colors, regardless of the color set for the layer.

TUTORIAL: CHANGING COLORS

- I. To change colors, start the **COLOR** command with one of these methods:
 - In the ribbon’s tab, click the Color Control droplist on the Properties panel, and then choose **Select Color**.
 - Or, at the ‘Command:’ prompt, enter the **color** command:
Command: **color** (Press ENTER.)
 - Alternatively, enter the aliases **col**, **colour** (the British spelling), or **ddcolor** (an old name) at the ‘Command:’ prompt.

2. In all cases, AutoCAD displays the Select Color dialog box.



3. Select a color sample (one of the colored squares), or enter a color name/number.
4. Click **OK**. All the objects you now draw take on the new color — until you change color again.

Note The Select Color dialog box is used by more than just the **COLOR** command; it is also accessed by any other command that sets colors, such as **LAYER**, **OPTIONS**, and **PROPERTIES**.

(*History:* Some drafters were upset when Autodesk introduced the **COLOR** command with AutoCAD v2.5. They felt that layer settings should always determine the colors of objects. Like this book's technical editor, these drafters feel it is wrong to override the layer's color setting with the **COLOR** command. Indeed, overriding colors is a poor method of working with CAD, which you may come across in your place of work. AutoCAD 2008 partially solved the problem by adding the **SETBYLAYER** command, which forces objects to take the color, linetype, lineweight, and plot style of their assigned layers.)

UNDERSTANDING AUTOCAD'S COLOR SYSTEM

AutoCAD lets you specify colors by a number of methods: select a color swatch from the dialog box, specify a number between 1 to 255, enter an RGB (red, green, blue) value, select a color book name and number, or type a color name such as “blue” for the first seven colors. By the way, *hue* refers to differences in a color, while *shade* refers to differences in gray.

(*History:* In early releases of AutoCAD, color was used to control pens in plotters. The color red, for example, could be assigned to pen #1, which in turn could be a red-colored pen or a black pen with a width of 0.1". Today in AutoCAD, you can control the plotter by color or with *styles*. More details follow in Chapter 20, “Plotting Drawings.”)

Color Numbers and Names

Of the 16.7 million colors available on computers, AutoCAD identifies the first seven by name and number, and the first 255 by number. Color numbers are called the “AutoCAD Color Index,” or **ACI** for short. Beyond that, colors are identified by RGB value (amounts of red, green, and blue) or by color book number.

The first seven color names and associated ACI numbers are listed below in numerical order:

ACI	Name	Abbreviation*
1	Red	R
2	Yellow	Y
3	Green	G
4	Cyan	C
5	Blue	B
6	Magenta	M
7	White (black)	W

* These abbreviations are used at the command line during commands like -COLOR, CHANGE, and -LAYER.

You may be wondering about my reference to the *first* 255 colors? Before AutoCAD 2004, the program worked with just 255 colors. These are the ones you see in the Index Color tab of the Select Color dialog box. (Colors 250 through 255 are shades of gray.) Every CAD package adopts a different order for colors, and so Autodesk identified AutoCAD’s as the ACI — *AutoCAD* Color Index.

Background Color 0

Color 0 is a special ACI color number that you never use in drawings. It identifies the *background* color of the drawing area (a.k.a viewport color). The background can be any color, but typically is white or black; AutoCAD 2010 uses a pale yellow (RGB = 254,252,240).

If you want to change the background color, use the dialog box accessed through the **A | Options | Display | Colors** menu sequence. This dialog box lets you change the colors of the background of the model layout, layout viewports, plot preview, block editor, and 3D projections views, as described in Chapter 3, “Setting Up Drawings.”

Color 7

Because white lines are nearly invisible against the pale yellow background, AutoCAD automatically changes color 7 (white) to the opposite of the background color (color #0) — here is the time in life when white is black, and black is white.

Officially, however, color 7 is white, because AutoCAD’s original background color was black. For this reason, you may encounter color 7 being referred to as “white,” even though it appears black on your screen, as well as black when plotted on paper.

Colors ByLayer, ByBlock, and ByEntity

In addition to the special colors numbers 0 and 7, AutoCAD has three more special “colors”: the oft-seen ByLayer, less-commonly seen ByBlock, and rarely-seen ByEntity. These solve the problem of what to name colors when they are controlled by layers, blocks, and entities (the old term for objects), respectively.

ByLayer Color #256

ByLayer means that the color of objects is determined by the layer’s color. Choosing ByLayer causes subsequently drawn objects to inherit the layer’s color. For example, if a layer is set to red, then all objects drawn on that layer are colored red automatically. (Layers are discussed later in this chapter.). Color ByLayer is also known as color #256.

ByBlock Color #0

ByBlock means that objects in blocks take on the color of the block. Choosing ByBlock causes objects to be drawn in white until they are turned into a block. When the block is inserted, the objects inherit the color of the block insertion. This color is also known as entity color #0.

ByEntity Color #257

ByEntity means that each object carries its own color, rather than one assigned by the layer or block. ByEntity is only used by system variables, such as **OBSCUREDCOLOR** and **INTERSECTIONCOLOR**, although programmers can use it in their software programs. This color has a value of 257.

True Color

AutoCAD now displays millions of colors. Technically, computer software works with three sets of 8-bit colors, which translates into 16.7 million hues (a.k.a. 24-bit color). In addition, another eight bits are used by today's graphics boards to display transparent objects; this is known as 32-bit color. (AutoCAD does not yet support transparency of objects, but Windows 7 and Vista do.)

It gets frustrating selecting a specific color out of millions of choices, and so AutoCAD provides several methods to help you make your selection. The methods are through RGB, HSL, and color books.

RGB

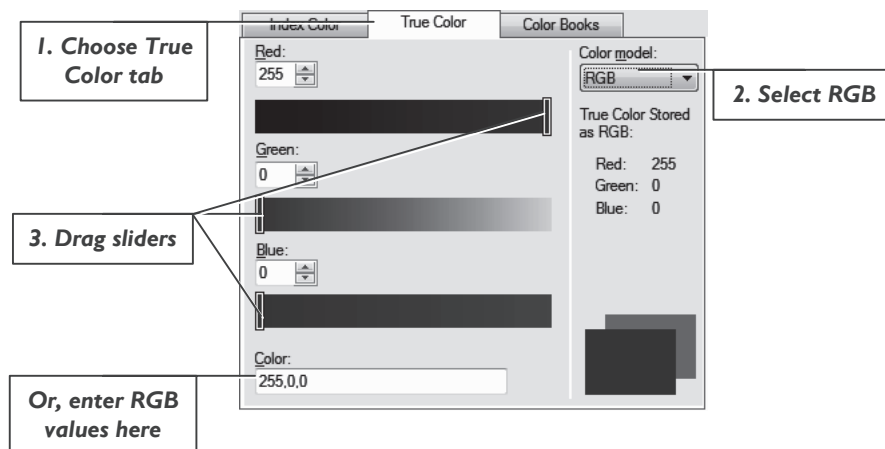
RGB is short for “red, green, blue,” the most common system for selecting colors. It specifies three sets of numbers that range from 0 to 255. Each number represents the amount of red, green, or blue, ranging from black (0) to full color (255).

As you may recall from elementary school physics, all colors of light are represented by varying amounts of red, green, and blue. Yellow is made from mixing red and green, for instance, and orange is made from full red (255) and half green (127), and no blue (0).

The hue of **blue** used in this book is represented by R=38, G=133, and B=187. Black consists of no colors (0,0,0), and white is all colors (255,255,255).

To specify a color in AutoCAD with the RGB system, follow these steps:

1. In the Select Color dialog box, select the **True Color** tab.
2. Under Color Model, select **RGB**.
3. Drag the sliders for Red, Green, and Blue.



Alternatively, enter an R,G,B color triplet, (such as 120,222,217) in the **Color** text box.

4. Click **OK**.

HSL

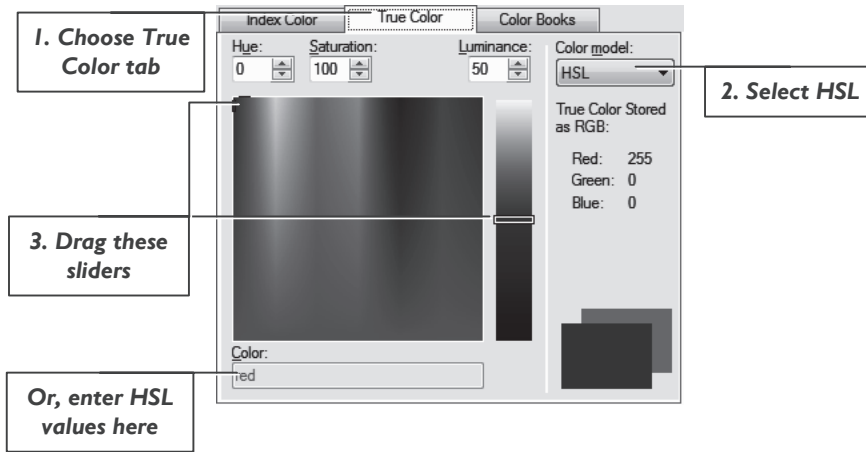
HSL is short for “hue, saturation, luminance.” It is a second system for specifying colors, which varies the *hue* (color), *saturation* (amount of color ranging from gray to full color), and *luminance* (brightness of color ranging from darkened to bright).

Hue numbers range from 0 to 360, starting at 0 = red. The colors progress through the colors of the rainbow: yellow is 60, green is 120, cyan is 180, blue is 240, violet is 300, and then red again at 360.

Saturation and luminance each range from 0 to 100 percent, where 0 is none (black) and 100 is full (white). The HSL value for the **blue** used in this book is H=202, S=66, and L=44.

To specify a color through HSL:

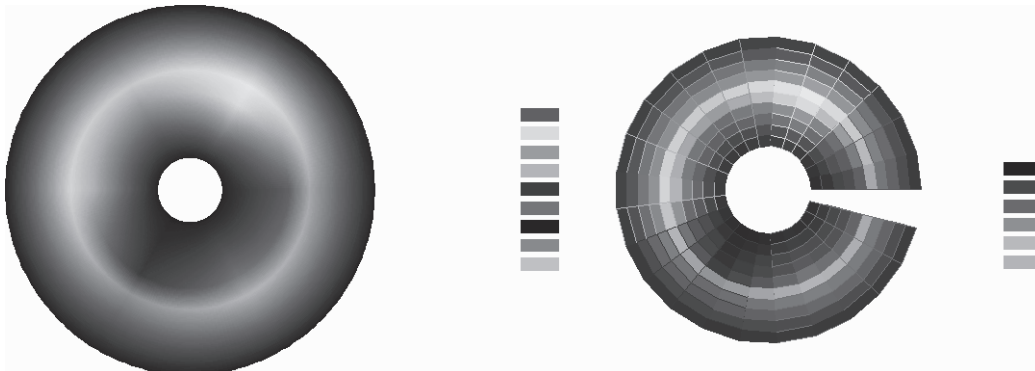
1. In the Select Color dialog box, select the **True Color** tab.
2. Under Color Model, select **HSL**.



3. Move the cross slider around to select the hue and saturation. Move the horizontal slider up and down to choose luminance.
Alternatively, In the Color text box, enter a color triplet, such as 120,222,217.
4. Click **OK**.

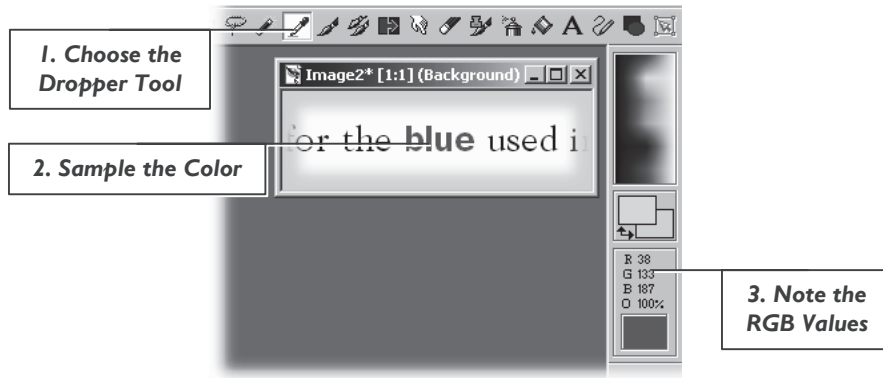


Note AutoCAD includes a drawing showing its colors in *color wheels*. In the `\autocad 2010\samples` folder, open `colorwh.dwg`. On the left, looking like a smoothly shaded donut, is the true color wheel made of 35,840 tiny tiles. (The coloring makes it appear like a 3D torus.)



On the right is the original 256-color ACI color wheel. It is flanked on the left by the seven named colors and on the right by six shades of gray.

RGB and HSL let you set precise colors in your drawings. For example, you can use the color of a client's logo.



Here's how I do it: get a sample of the color, either by grabbing a screen or by scanning a page. Open the sample in a raster editor like PaintShop Pro. Use the color dropper tool to sample the color; note the RGB values, and then enter the three numbers into AutoCAD's Select Color dialog box.

Color Books

AutoCAD provides a third naming system used by the publishing and design industries, called "color books." Colors are seen differently by individuals, and so a standardized system for specifying a select number of colors was needed. Color books consist of swatch names and numbers.

For example, the name of the **blue** color used in this book is Pantone 2925. When I specify this name, the identical hue of blue shows up in my desktop publishing software, in the printed book, and in AutoCAD.

AutoCAD supports three of these systems: the American Pantone, the European RAL, and the Japanese DIC systems.

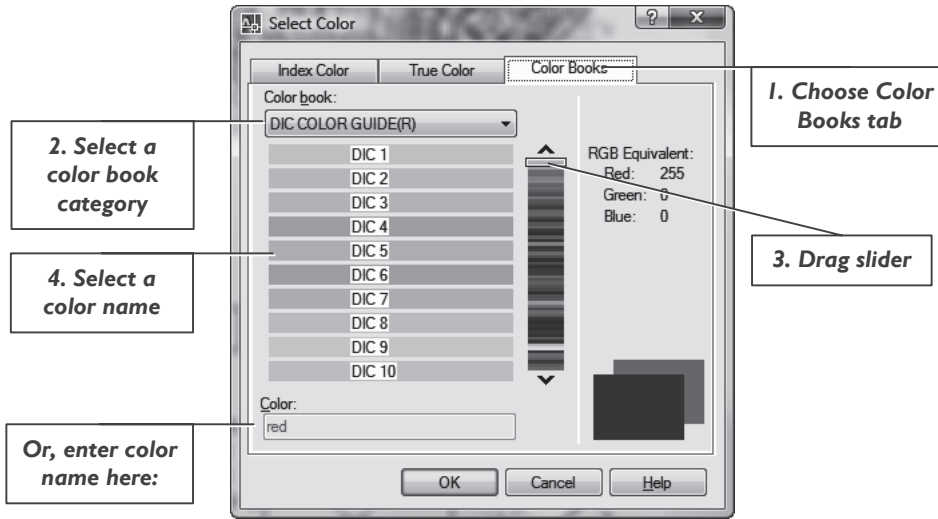
The Pantone Matching System (www.pantone.com) was designed in 1963 to specify colors for the graphic arts, textiles, and plastics industries. Today, designers typically work with Pantone's fan-format book of standardized colors. This system is used primarily in North America.

The RAL color system (www.ral.de) was designed back in 1927 to standardize colors by limiting the number of color gradations, first to just 30 and now to over 1,600. RAL (Reichs Ausschuß für Lieferbedingungen – German for "Imperial Committee for Supply Conditions") is administered by the German Institute for Quality Assurance and Labeling. This system is used primarily in Europe.

The DIC Color Guide (www.dic.co.jp, Japanese language only) is the Japanese standard for specifying colors. DIC is the abbreviation for the ink company that came up with the system, Dainippon Ink and Chemicals.

Here's how to specify colors by Color Books in AutoCAD:

1. In the Select Color dialog box, choose the **Color Books** tab.



2. Under **Color Books**, select a DIC, Pantone, or RAL book name.
3. Move the slider up and down to choose a color group, and then pick a specific color name. As an alternative, in the Color text box, enter a color book specification.
4. Click **OK**.

CHANGING COLORS: ADDITIONAL METHODS

AutoCAD provides another way to change colors through the **Color Control** droplist in the Properties panel of the ribbon. The control affects color by two methods, depending on the order in which you use them:

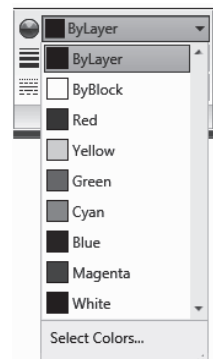
Set the working color (subsequent objects taking on the color) — select a color from the droplist.

Change the color of object(s) — first select the objects, and then a color from the droplist.

In the following tutorial, we look at both methods.

TUTORIAL: SETTING AND CHANGING COLORS

1. Start AutoCAD with a new drawing, and then draw a few lines. Notice that they are colored white or black — depending on the background color of your screen.
2. From the ribbon's Properties panel, click on the **Color Control** droplist. Notice the list of basic colors: ByBlock, ByLayer, and the first seven colors, red through white.
3. Select **Red**.
4. Draw a few more lines. Notice that they are colored red.
5. Now change the red lines to blue: select the red lines by picking them. They become highlighted and show grips (squares).
6. From the Color Control droplist, select **Blue**. Notice that the lines turn blue.
7. Press **ESC** to “unselect” the lines (remove the highlighting and grips).



LWEIGHT

The **LWEIGHT** command (short for “lineweight”) changes the visual width of lines and other objects.

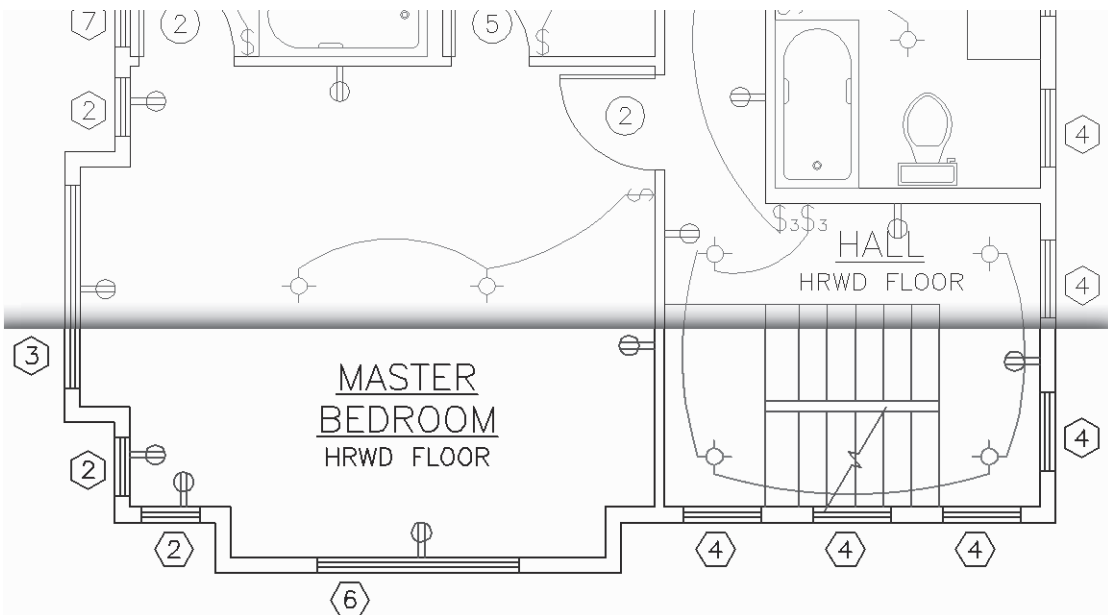
(History: In old releases of AutoCAD, only polylines and traces could have a width (weight); all other objects were drawn one pixel wide, as thin as the display. Widths were assigned at plot time based on the object’s color.)

Using a variety of weights (heavy and thin lines) helps make drawings clearer. Weights can be assigned to layers or to individual objects. In the figure below, the upper part of the drawing has lineweights turned off; in the lower part, lineweights are turned on.

Lineweights range from 0.05 mm (0.002") to 2.11 mm (0.083"). AutoCAD displays the values in millimeters or inches. The lineweight of 0 is compatible with earlier versions of AutoCAD, and displays as one pixel in model space. It is plotted at the thinnest width of which the plotter is capable.

When on, the weight appears when both displayed and plotted. The display is constant, in pixels. This means that as you zoom out, the lines appear to get wide — WYSINQWYG — “what you see is not quite what you get,” in the words of the technical editor.

There is an exception: objects with lineweights under a certain value are displayed one pixel wide in model space. The default is 0.025mm but can be adjusted by the Display Scale slider in the Linetype Settings dialog box, described below.



At plot time, objects with weight are plotted at the exact same width. Objects copied to the Clipboard (through CTRL+C) retain their lineweight data when pasted (CTRL+V) back into AutoCAD, but may lose their weight when pasted into other Windows applications.

Some drafters prefer to define lineweight through layers, as with colors; they do not approve of overriding the layer’s lineweight ByLayer setting.

AutoCAD includes three named lineweights:

ByLayer — objects take on the lineweight defined by the layer.

ByBlock — objects take on their block’s lineweight.

Default — displays the default value of 0.25 mm, or another default value.



Note Lineweights are meant as a visual aid. Do not use lineweights to represent the actual width of objects. If printed circuit board traces are 0.05 inches wide, use polylines with width of 0.05 inches; don't draw lines with a weight of 0.05 inches.

TUTORIAL: CHANGING LINEWEIGHTS

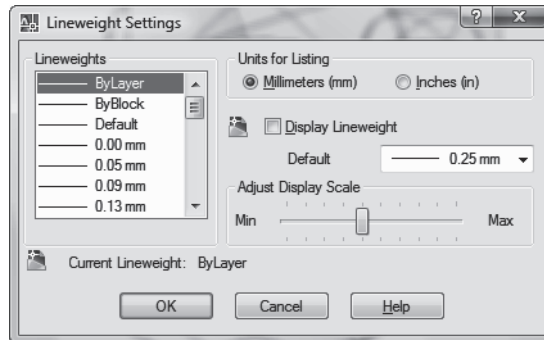
- 1. In new drawings, lineweights are not displayed. To turn on their display, choose the **LWT** button on the status bar. Lineweights are on when the button looks blue.
- 2. To change the lineweights of objects, start the **LWEIGHT** command with one of these methods:
 - In the ribbon's Home tab, click the lineweight droplist in the Properties panel.

AUTOCAD'S DEFAULT LINEWEIGHTS

The table below shows the default lineweight values used by AutoCAD and their equivalent values for industry standards. There are 25.4 mm per inch; 72.72 points per inch.

Millimeters	Inches	Points	Pen Size	ISO	DIN	JIS	ANSI
0.50	0.002						
0.90	0.003	1/4 pt					
0.13	0.005				✓		
0.15	0.006						
0.18	0.007	1/2 pt	0000	✓	✓	✓	
0.20	0.008						
0.25	0.010	3/4 pt	000	✓	✓	✓	
0.30	0.012		00				2H or H
0.35	0.014	1 pt	0	✓	✓	✓	
0.40	0.016						
0.50	0.020		1	✓	✓	✓	
0.53	0.021	1 1/2 pt					
0.60	0.024		2				H, F, or B
0.70	0.028	2 1/4 pt	2 1/2	✓	✓	✓	
0.80	0.031		3				
0.90	0.035						
1.00	0.039		3 1/2	✓	✓	✓	
1.06	0.042	3 pt					
1.20	0.047		4				
1.40	0.056			✓	✓	✓	
1.58	0.062	4 1/4 pt					
2.00	0.078			✓	✓		
2.11	0.083	6 pt					

- On the status line, right-click **LWT**, and then choose **Settings** from the shortcut menu.
 - At the 'Command:' prompt, enter the **lweight** command:
Command: **lweight** (Press ENTER.)
 - Alternatively, enter the aliases **lw** or **lineweight** at the 'Command:' prompt.
3. In all cases, AutoCAD displays the Lineweight Settings dialog box.



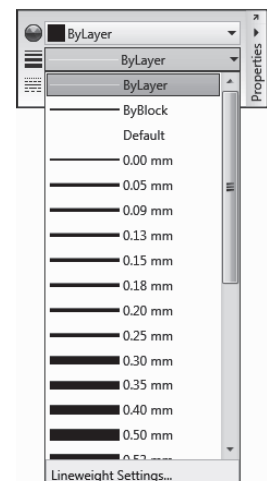
4. Under Lineweights is the list of weights available in AutoCAD. (This is the same list displayed by the Lineweights droplist in the Properties panel. You cannot customize lineweights.)
Select a lineweight; this becomes the default until you change the lineweight again.
5. Other items of interest in the dialog box include:
- Units for Listing** — selects between inches and millimeters (the default).
- Display Lineweight** check box — functions identically to the **LWT** button on the status bar.
- Default** list box — specifies the lineweight with which all objects are drawn, unless overridden by layer.
- Adjust Display Scale** slider — controls the display scale of lineweights in model space. This slider is beneficial if your computer displays AutoCAD on a high-resolution monitor, which tends to make lines look thinner. Experiment with adjusting the lineweight scale to see if you get a better display with different lineweights. As you move the slider, notice that the widths of lines in the Lineweights list change in clusters of 4-5 weights, and not uniformly.
6. Choose **OK** to dismiss the dialog box.

Lineweight Control

As an alternative to using commands and dialog boxes, you can change lineweights directly through the Lineweight Control droplist. As with color, this control affects lineweights by two methods, depending on the order in which you use it:

Set the working lineweight (subsequent objects taking on the selected lineweight): first select a lineweight from the droplist, and then draw.

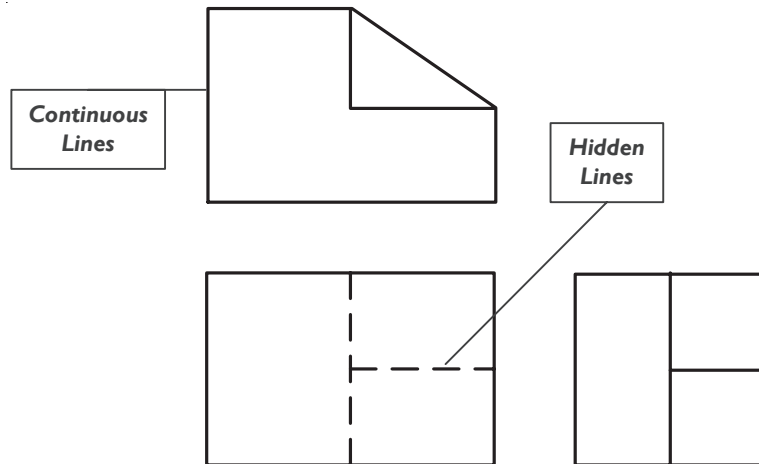
Change the lineweight: first select the objects, and then a lineweight from the droplist.



LINETYPE

The **LINETYPE** command sets the patterns of lines.

Linetypes are used to identify objects in 2D drawings. For example, solid lines represent the edges of objects; AutoCAD refers to solid lines as “continuous lines.” Lines marked with “HW” represent hotwater pipes.



Edges hidden from view are shown in a linetype referred to as “hidden,” constructed of short segments. The figure shows an object containing edges that are hidden in some views. These edges are defined with the Hidden linetype.

AutoCAD provides many linetypes, and you can create custom ones. AutoCAD employs three classes of linetype:

Simple — consist of line segments, dots, and gaps; these are the most commonly-used linetypes.

ISO — resemble simple linetypes, except that they conform to standards set by the International Organization of Standards, and can have pen widths assigned to them.

Complex — use dashes and spaces like simple linetypes, but add characters and shapes, such as HW (mentioned above) or squiggles (to indicate insulation).

AutoCAD stores linetype definitions in *acad.lin* and *acadiso.lin*, and the shapes for complex linetypes in *ltypeshp.shp* — all found in the *\support* folder. You can create your own linetypes, as well.

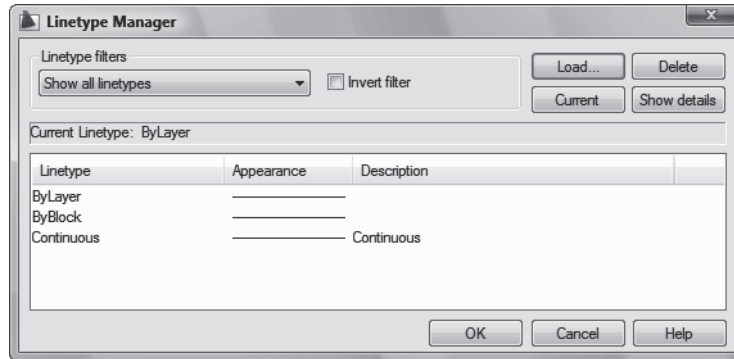
Using linetypes in drawings takes two steps: (1) loading them into the drawing, and (2) selecting them for use. Applying them by layer is the preferred method.

TUTORIAL: CHANGING LINETYPES

- I. To change the linetype, start the **LINETYPE** command with one of these methods:
 - In the ribbon's tab, click the **Linetype** droplist on the Properties panel, and then choose **Other**.
 - At the 'Command:' prompt, enter the **linetype** command:

Command: **linetype** (Press ENTER.)
 - Alternatively, enter the aliases **lt**, **ltype**, or **ddltype** (the old name) at the 'Command:' prompt.

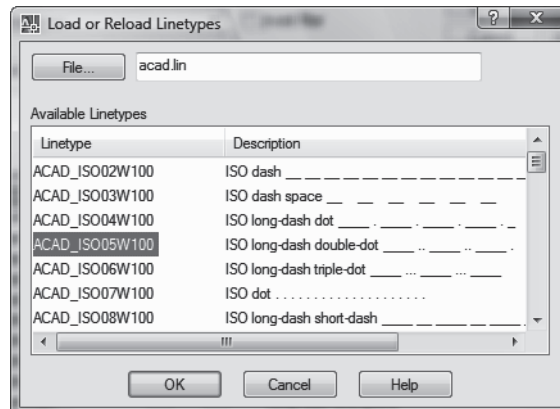
2. In all cases, AutoCAD displays the Linetype Manager dialog box:



The dialog box lists the linetypes loaded into the current drawing. New drawings have three: Continuous and the same two special names that appear in colors and lineweights:

- ByLayer** — assigned by layers.
- ByBlock** — assigned by blocks.

3. To load additional linetypes, choose the **Load** button. AutoCAD displays the Load or Reload Linetypes dialog box. (Unless you know you have linetype definitions stored in a file other than *acad.lin*, you can ignore the File button.)



4. I find it easier to load all linetypes at once, and then later remove any that remain unused. To do so, right-click on any linetype, and then choose **Select All**. AutoCAD highlights all linetypes.
5. Choose **OK**, and AutoCAD loads all linetypes.
6. Back in the Linetype Manager dialog box, select a linetype, and then click **Current**.
7. Choose **OK**.

From now on, all objects are drawn in this linetype — until you change it again.

Global, Local, and Annotative Linetype Scales

AutoCAD lets you specify linetype scales in three ways:

- **Globally** for all objects in the drawing. The global linetype scale is set by the **LTSCALE** command, described later.
- **Locally** for individual objects. Although it is poor drafting practice to use more than one linetype scale in drawings, local scaling is sometimes needed for complex linetypes, such as Batting and Hot Water Supply.

Linetype	Description
ACAD_ISO02W100	ISO dash
ACAD_ISO03W100	ISO dash space
ACAD_ISO04W100	ISO long-dash dot
ACAD_ISO05W100	ISO long-dash double-dot
ACAD_ISO06W100	ISO long-dash triple-dot
ACAD_ISO07W100	ISO dot
ACAD_ISO08W100	ISO long-dash short-dash
ACAD_ISO09W100	ISO long-dash double-short-dash
ACAD_ISO10W100	ISO dash dot
ACAD_ISO11W100	ISO double-dash dot
ACAD_ISO12W100	ISO dash double-dot
ACAD_ISO13W100	ISO double-dash double-dot
ACAD_ISO14W100	ISO dash triple-dot
ACAD_ISO15W100	ISO double-dash triple-dot
BATTING	Batting
BORDER	Border
BORDER2	Border (.5x)
BORDERX2	Border (2x)
CENTER	Center
CENTER2	Center (.5x)
CENTERX2	Center (2x)
DASHDOT	Dash dot
DASHDOT2	Dash dot (.5x)
DASHDOTX2	Dash dot (2x)
DASHED	Dashed
DASHED2	Dashed (.5x)
DASHEDX2	Dashed (2x)
DIVIDE	Divide
DIVIDE2	Divide (.5x)
DIVIDEX2	Divide (2x)
DOT	Dot
DOT2	Dot (.5x)
DOTX2	Dot (2x)
FENCELINE1	Fenceline circle
FENCELINE2	Fenceline square
GAS_LINE	Gas line
HIDDEN	Hidden
HIDDEN2	Hidden (.5x)
HIDDENX2	Hidden (2x)
HOT_WATER_SUPPLY	Hot water supply
JIS_02_0.7	HIDDEN0.75
JIS_02_1.0	HIDDEN01
JIS_02_1.2	HIDDEN01.25
JIS_02_2.0	HIDDEN02
JIS_02_4.0	HIDDEN04
JIS_08_11	1SASEN11
JIS_08_15	1SASEN15
JIS_08_25	1SASEN25
JIS_08_37	1SASEN37
JIS_08_50	1SASEN50
JIS_09_08	2SASEN8
JIS_09_15	2SASEN15
JIS_09_29	2SASEN29
JIS_09_50	2SASEN50
PHANTOM	Phantom
PHANTOM2	Phantom (.5x)
PHANTOMX2	Phantom (2x)
TRACKS	Tracks
ZIGZAG	Zig zag

- **Annotatively** for scale-specific viewports.

Local (Object) Linetype Scaling

To change the linetype scale of individual objects, use either the **PROPERTIES** command (as described later in this chapter) or the **CELTSCALE** system variable (short for “current entity linetype scale”). There is a difference between the two: the **PROPERTIES** command changes the linetype scale of objects that already exist in the drawing, while the **CELTSCALE** system variable changes the scale of objects you plan to draw.

CELTSCALE can be tricky to use, because it sets the linetype scale *relative* to the value set by the **LTSCALE** command setting: when **LTSCALE** is set to 3.0 and **CELTSCALE** is set to 0.75, objects will be drawn with a linetype scale of 2.25 (3.0 x 0.75 = 2.25).

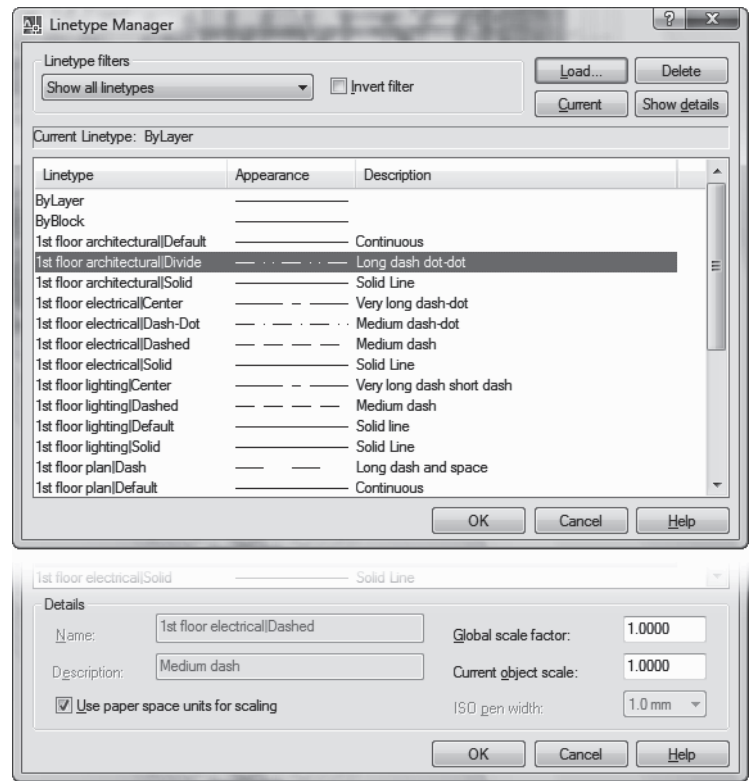
Both global and object linetype scaling are available as features “hidden” in the Linetype Manager dialog box. Click the **Details** button to reveal them.

Global scale factor — sets the linetype scale for all objects in the drawing. This scale factor takes effect with the next drawing regeneration.

Current object scale — sets the linetype scale for subsequently-created objects.

ISO pen width — selects from one of the ISO’s pen widths, such as 1.00 mm. This option is grayed out when non-ISO linetypes are selected. To activate, double-click an ISO linetype.

Use paper space units for scaling — scales linetypes identically in paper space and model space through the **PSLTSCALE** system variable.



Annotative Scaling

Annotative scaling makes linetypes appear at the correct size in model space viewports. This includes the Model tab and viewports in Layout tabs. When a scale factor is assigned a viewport (or in Model tab), then the scale factor for all linetypes is adjusted automatically by AutoCAD.

The **MSLTSCALE** system variable (short for “model space linetype scale”) toggles annotative scaling of linetypes. When set to 1, linetypes are scaled to match the current viewport scale; this is the default setting. (The variable is set to 0 in drawings opened from AutoCAD 2007 and earlier.)

To use **MSLTSCALE** correctly, you have to set all linetype scale system variables to 1:

CELTSCALE = 1
LTSCALE = 1 (or another size correct for plotting)
MSLTSCALE = 1
PSLTSCALE = 1

The changes due to annotation scaling appear with the next regeneration (**REGEN** command).

LINETYPE SCALING (LTSCALE)

Because linetypes are constructed of line segments and dashes, the pattern can look too small or too large for drawings. How do you tell when the scale is wrong? When the pattern lines do not look continuous.

Like hatch patterns, linetypes need to be set to an appropriate scale; fortunately, the same scale factor applies to both hatch patterns and linetypes. When you figure out the correct factor for one, you can use it for the other, as well as for text and dimensions.

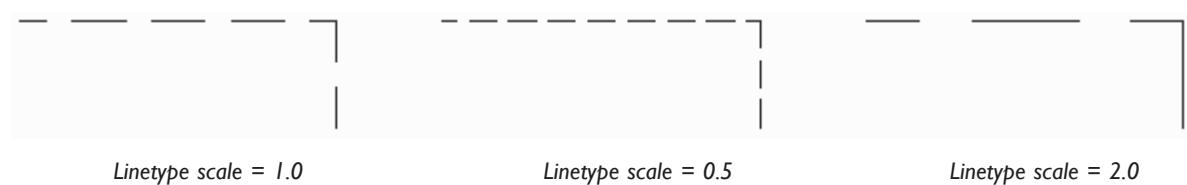
The scale of linetypes is adjusted by the **LTSCALE** (short for “linetype scale”) command:

Command: **ltscale**

Enter new linetype scale factor <1.0000>: *(Enter a value.)*

Regenerating model.

Numbers larger than 1.0 result in longer line segments, while numbers smaller than 1.0 create shorter line segments.



To display the new linetype scale correctly, AutoCAD automatically performs a regeneration. If a regeneration does not occur after you change the linetype scale, force a regeneration with the **REGEN** command.

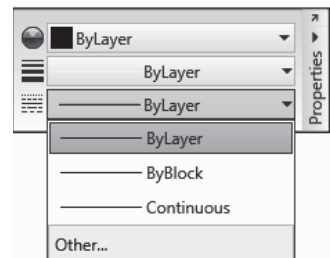
If, after changing the scale, the linetypes still appear continuous, change **LTSCALE** to yet a different value. It is possible for the linetype scale to be too large or small to display. If lines take longer to draw, it is likely that the scale is too small.

General rule: the linetype scale is the inverse of the plot scale. For example, when the plot scale is $1/8" = 1'$ (1:96), the linetype scale factor is 96. The exception is when your drawing uses annotative scale factor, in which case AutoCAD 2008 (and later) uses the viewport scale for linetypes when **MSLTSCALE** = 1 (the default setting).

Linetype Control

As with color and lineweights, you can change linetypes directly through the ribbon's **Linetype Control** droplist. The control affects linetypes by two methods, depending on the order in which you use it:

- Set the working lineweight (objects subsequently taking on the selected linetype) — select a linetype name from the droplist.
- Change the linetype of existing object(s) — first select them, and then a linetype name from the droplist.





Note If you (or someone else) creates custom linetypes, AutoCAD has two ways to share them. One is to exchange a copy of the .lin file that stores the custom definitions. The other is to use DesignCenter to drag a copy of the linetype from one drawing to another.



PROPERTIES

The **PROPERTIES** command displays a palette for changing (almost) *all* properties of objects.

The Properties palette displays different sets of properties depending on the object(s) you select: text, mtext, 3D face, 3D solid, multiline, arc, point, attribute definition, polyline, block insertion, ray, body, region, circle, shape, dimension, mleader, 2D solid, ellipse, spline, external reference, hatch, tolerance, image, trace, leader, viewport, line, xline, and more.

That's because each object has a somewhat different set of modifiable properties. For example, you can modify the endpoints of a line through x, y, and z coordinates; you cannot, however, modify the endpoint of an arc; instead, you have to alter its start and end angle.

TUTORIAL: CHANGING ALL PROPERTIES

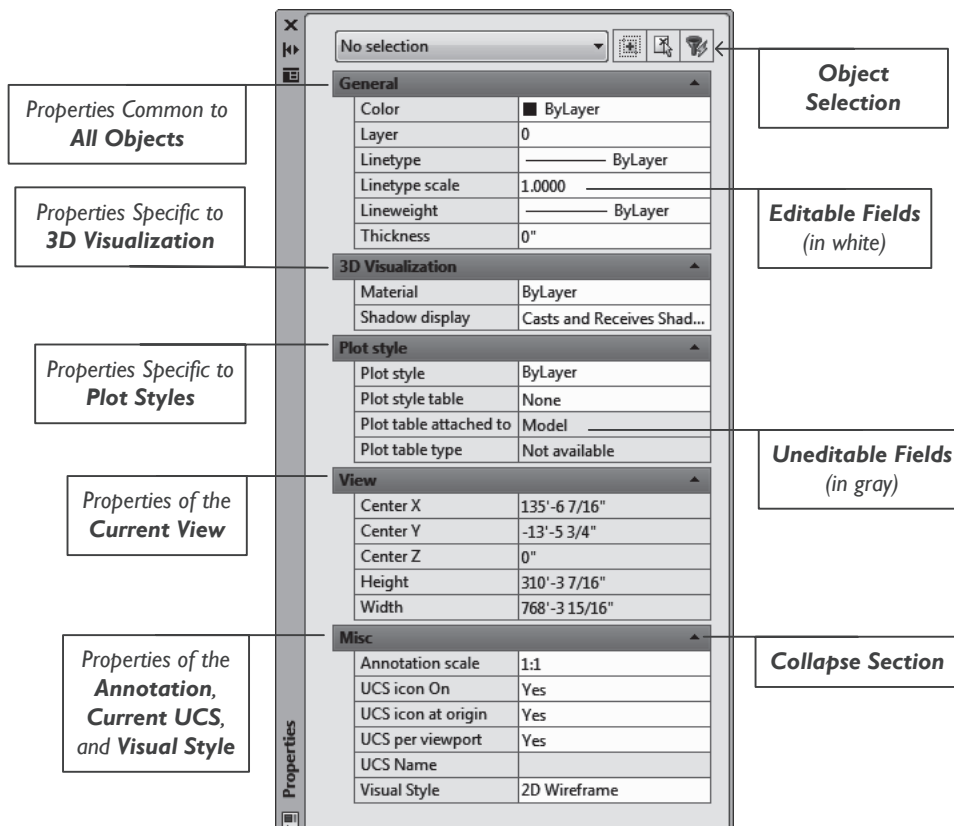
1. To change all the properties of objects, start the **PROPERTIES** command with one of these methods:

- At the keyboard, press **CTRL+I**.
- At the 'Command:' prompt, enter the **properties** command:

Command: **properties** (Press ENTER.)

- Alternatively, enter the aliases **pr**, **props**, **mo** (short for "modify"), **ch**, (short for "change"), **ddchprop**, or **ddmodify** (its old name) at the 'Command:' prompt.

In all cases, AutoCAD displays the Properties palette (see figure, next page).



Notice that there are two colors of field: white and gray. *White* means you can modify the property, while *gray* means you cannot.

The effect of changes you make in this window depends on whether or not objects are selected — much like changing settings in the Properties panel.

- When no objects are selected (as shown above), changing the properties affects objects drawn from now on.
- When one or more objects are selected, changing the properties affects the selected objects immediately.

You can collapse unneeded sections by clicking the double button, which makes the palette smaller.

Notice the section named “General” located in the upper part of the window. The General section includes these modifiable properties:

Color — selects the color square to change colors.

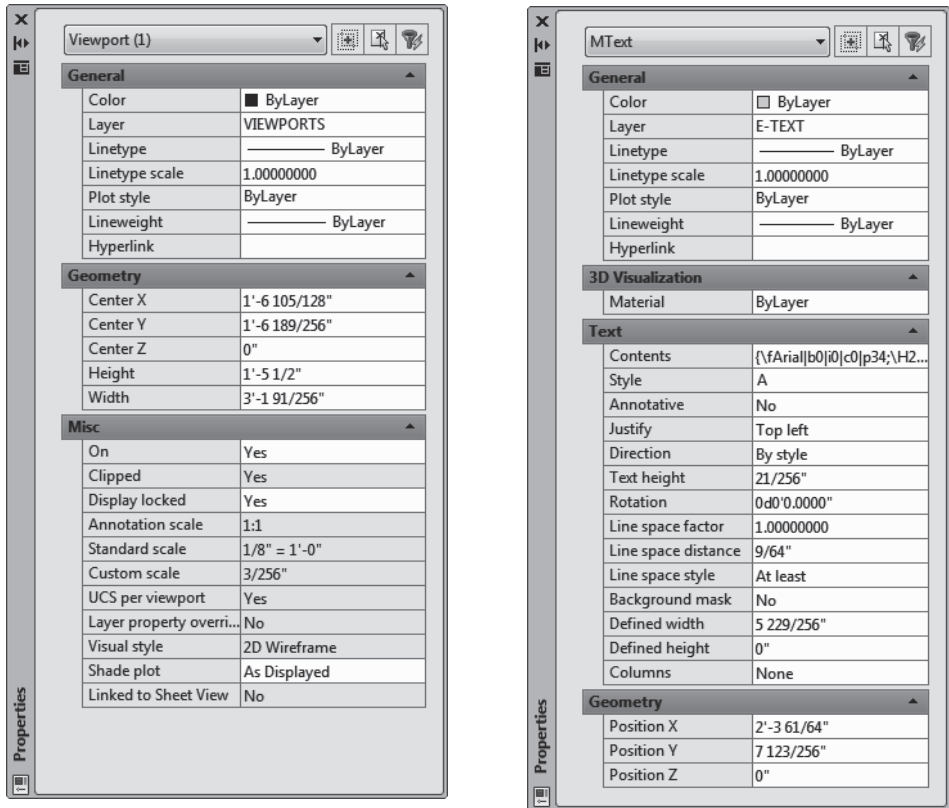
Layer — selects a layer name.

Linetype — selects a linetype name.

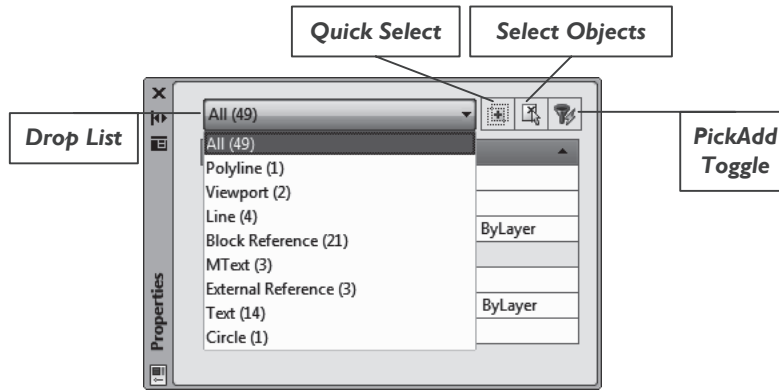
Linetype Scale — specifies a different scale factor.

Lineweight — selects a preset lineweight.

2. To inspect the properties of one or more objects, pick them in the drawing. Note that you do not dismiss the Properties palette: simply move the cursor into the drawing, and then pick the objects. (To select all objects in the drawing, press **CTRL+A**.) Notice that the palette changes its content to reflect the properties of the selected object(s), such as the viewport at left, and the mtext at right:



As an alternative to picking, choose one of the buttons at the top of the palette:



The droplist shows the variety (and, in parentheses, the number) of objects.

Quick Select displays the Quick Select dialog box, described in Chapter 7, “Drawing with Efficiency.”

Select Objects causes AutoCAD to prompt you (unnecessarily):

Select objects: *(Select one or more objects.)*

Select objects: *(Press ENTER to end object selection.)*

Toggle value of PICKADD sysvar changes how you select objects:

- Plus (+) sign means you select additional objects by picking them.
- Number 1 means you select additional objects by holding down the **SHIFT** key.

You can modify any property whose value has a white background.

3. In the Geometry section, click a field, such as **Height**.
4. Change the value to a different number. Notice that the geometry changes in the drawing.

Geometry	
Center X	1'-6 105/128"
Center Y	1'-6 189/256"
Center Z	0"
Height	1'-5 1/2"
Width	3'-1 91/256"

As alternatives:

- Click the  button found on the toolbar. AutoCAD prompts you:
Pick a point in the drawing: *(Move the cursor, and then pick a point.)*
Use object snaps to make accurate picks.
- Click the calculator icon to bring up **QuickCalc**.

5. When done with this palette, you can click the **x** or enter the **PROPERTIESCLOSE** command.



Note You can easily display the Properties window by double-clicking (almost) any object in the drawing. Exceptions include double-clicking text, which displays the text editing window.

CHANGING PROPERTIES: ADDITIONAL METHODS

AutoCAD provides other ways to change properties:

- **QUICKPROPERTIES** command changes properties in a floating palette near the cursor.
- Rollover Tips display object properties in a tooltip.

QuickProperties

The QUICKPROPERTIES command toggles the display of a properties palette that floats near the cursor.

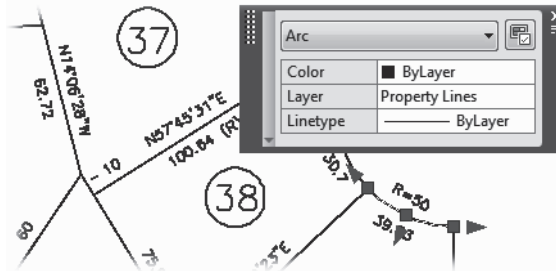
Command: **quickproperties**

<Quick properties on.>

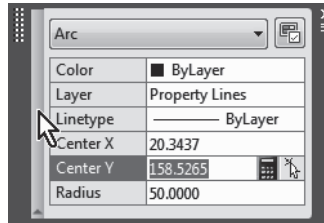
Enter the command a second time to turn off Quick Properties. Alternatively, press CTRL+SHIFT+P.

To use this palette, follow these steps:

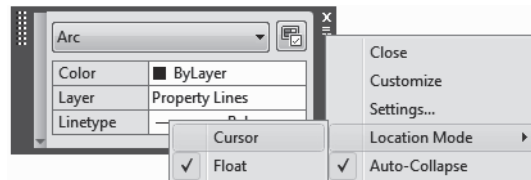
1. Ensure Quick Properties is turned on.
2. Choose an object in the drawing. Notice that the palette appears near the cursor.



3. Move the cursor over the palette. Notice that it expands to show additional properties.
4. As in the Properties palette, you can change the values of properties.



5. To change the settings of the palette, click the **Options** button under the **X**.



Close — closes the palette, but in a curious manner: the palette will no longer be displayed for the current object, an arc in this case. It will, however, be displayed for all other objects.

Customize — lets you choose the individual properties for every object type displayed by this palette. It displays the Customize User Interface dialog box.

Settings — sets display options, several of which are available in this menu. It displays the Quick Properties tab of the Drafting Settings dialog box.

Location Mode — determines where the palette appears: near the cursor, or floating in a fixed position. I find I prefer Float; otherwise, the palette obscures the drawing right where I am working.

Auto-Collapse — determines whether the palette becomes smaller when the cursor leaves its territory. I prefer this option turned off, since the collapsing and expanding action of the palette gets irritating.

QuickProperties Settings

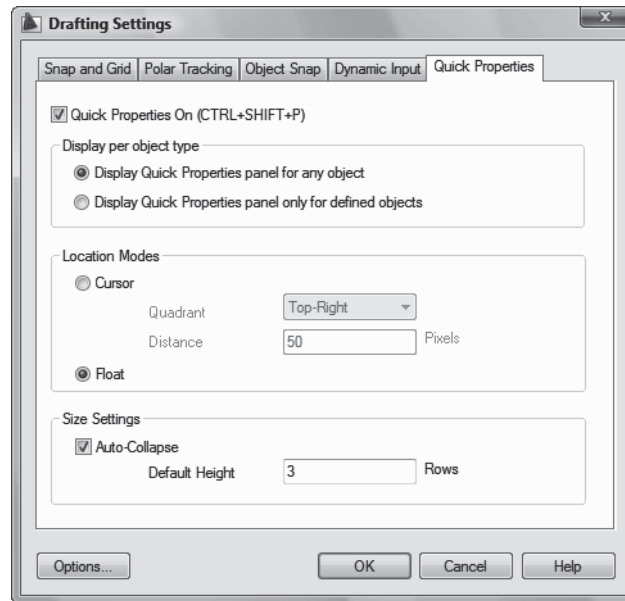
The Quick Properties tab of the **DSETTINGS** command provides the following options for controlling the placement of the Quick Properties palette:

Quick Properties On — toggles the display of the palette.

Display Per Object Type — determines if the palette is always displayed, or just for specified objects.

Location Modes — specifies where the palette appears: in a fixed position (called “Float” by Autodesk), or near the cursor. When near the cursor, you have the option of specifying the distance away from the cursor, as well as which cursor quadrant it appears in.

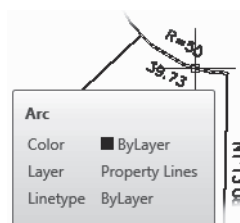
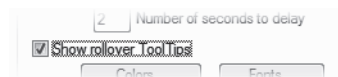
Size Settings — controls the default size of auto-collapsed palettes.



Rollover Tips

Rollover tips display object properties in a tooltip as the cursor passes over objects. To use rollover tips, follow these steps:

- I. Ensure rollover tips are turned on:
 - a. Enter the **OPTIONS** command.
 - b. Click the **Display** tab.
2. Move the cursor over an object. Notice the tooltip that appears.



Rollover tips differ from Quick Properties in the following ways. You do not select the object; just roll the cursor over objects. Rollover tips cannot edit object properties.

The content of the rollover tip is customizable through the CUI command's Rollover Tooltips mode. There you can choose among dozens of properties for 98 object types. In the case of arcs, you have 28 properties to choose from, ranging from the useful (Arc Length) to the obscure (Normal Y).



The **LAYER** command applies common sets of properties to objects assigned to layers.

Nearly all CAD drawings are constructed using *layers* that can be turned on and off, and changed. Layers go back to the days of traditional drafting, which sometimes involved a method of drawing called “overlay drafting.” Drafters overlaid sheets of semitransparent drafting media so that the drawings showed through each other. Multiple sheets could be printed together, resulting in prints that showed the complete design.

The bottom sheet was typically referred to as the “base drawing.” Each additional sheet showed different items. For instance, when a drafter prepared a set of floor plans, he typically drafted separate drawings showing the items to be removed, plumbing, electrical, and so on. Often, the floor plan is the base drawing, with each discipline, such as electrical and plumbing, placed on overlay sheets.

AutoCAD's layers are like sheets of glass stacked on top of one another. You place different aspects of the drawing on the sheets of glass (layers), yet are able to see through all the layers. The work appears as though it were one drawing.

AutoCAD goes two steps further. You can turn layers on or off, so that they are either visible or invisible, as well as change their properties. The properties of a layer affect all objects assigned to the layer, but can be overridden with other commands, such as **COLOR**, **LINETYPE**, and **PROPERTIES** (described earlier this chapter). In turn, these overrides can be overridden.

Before adding objects to a layer, you switch to the appropriate layer. Anything you now draw is placed on this layer.

The working layer is called the “current layer.” AutoCAD can have only one layer current at a time. The current layer cannot be *frozen*. (Frozen layers are invisible and cannot be edited.) A layer named “0” exists in all new drawings; it has some special properties, and cannot be renamed or removed.



Note The **LAYER** command displays layer information in a palette. Like other palettes, it is meant to be open all the time. As you change layer properties in the palette, they are reflected immediately in the drawing.

If you prefer to use the layer dialog box, enter the **CLASSICLAYER** command. Alternatively, if you want the **LAYER** command to show the dialog box, set the new **LAYERDLGMODE** system variable to 1. (You can still access the layer palette with the **LAYERPALETTE** command.)

TUTORIAL: CREATING LAYERS

- I. To create layers, start the **LAYER** command with one of these methods:
 - In the ribbon's 2D Home tab, click the **Layer Properties** button on the Layers panel.
 - Or, at the 'Command:' prompt, enter the **layer** command:
 Command: **layer** (Press ENTER.)
 - Alternatively, enter the aliases **la** or **ddlmodes** (an old name for this command) at the 'Command:' prompt.

Working with Layers

The Layer Properties Manager palette provides you with a fair degree of control over layers.

Status

The **Status** column uses icons to report the status of each layer:

Status	Comment
	Layer is current.
	Layer is in use (has at least one object).
	Layer is empty (can be deleted, unless part of an xref).
	Layer is marked for deletion.
	Layer is part of a group filter.
	Layer is part of a property filter.

 Turning Layers On and Off

When layers are *on*, their objects are seen and can be edited; the status is indicated by the yellow light bulb icon in the On column. When *off*, the objects on that layer are not seen; the light bulb icon turns blue-gray.

To turn one or more layers on or off, first highlight the layer(s) by selecting their name(s), and then choose the light bulb icon. The “on” column immediately reflects the change.

On and **Off** were used in the earliest versions of AutoCAD to toggle the display of layers, and are rarely used anymore; **Freeze** and **Thaw** are now preferred.

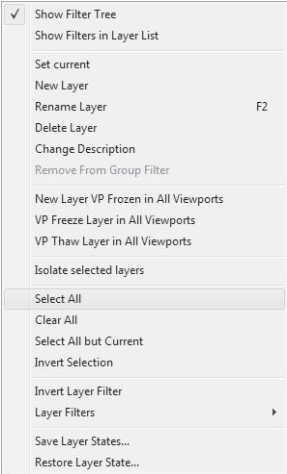
 Freezing and Thawing Layers

Frozen layers cannot be seen or edited; their status is shown by the snowflake icon in the **Freeze** column. *Thawed* layers can be seen and edited, just like *on* layers; the symbol for thawed layers is the sun icon.

To freeze or thaw one or more layers, highlight the target layer(s), and then choose the sun or snowflake icon. It is more efficient to freeze layers than to turn them off. Objects on frozen layers are ignored by AutoCAD during regenerations and other computationally-intensive operations.



Notes To select *all* layers, right-click any layer name to display a shortcut menu. Choose **Select All** to highlight all layer names. Choose **Clear All** to deselect all layers.



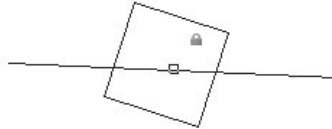
To change the width of the columns in the layer dialog box, grab the black bar separating column tiles and drag left or right.

Locking and Unlocking Layers

Objects on *locked* layers can be seen, but not edited in most cases. You can still use these objects for osnaps, trimming, and extending.

Locked layers show a padlock icon in the Lock column. *Unlocked* layers are like *on* layers: their objects are seen and can be edited; an unlocked layer shows an open padlock.

When you attempt to select objects on locked layers, AutoCAD shows a padlock icon near the cursor.



To lock or unlock layers, select the target layer(s), and then choose the padlock icon(s) in the Lock column.

■ Setting Layer Colors

To set the color for objects on layers, first select the layer name(s), and then click on the color square. AutoCAD displays the Select Color dialog box. Choose a color, and then click **OK**.

From now on, all objects on the layer are displayed in that color, unless overridden by the **COLOR** command or by a block's properties.

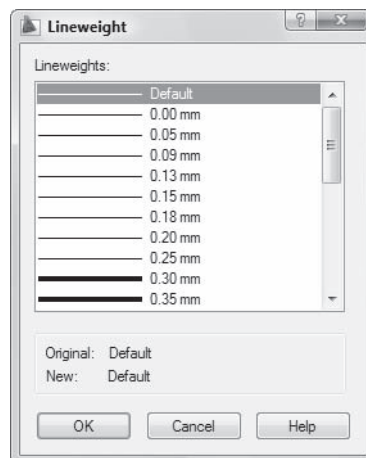
Setting Layer Linetypes

When you select a linetype for a layer, it affects all the objects on that layer. First select the layer name(s) for which you wish to set a linetype, and then choose the linetype. AutoCAD displays the Select Linetype dialog box. If necessary, load linetypes. Select a linetype and choose **OK**.

Setting Layer Lineweights

You can specify the lineweights for objects residing on layers.

1. Select the layer name(s) for which you wish to set a lineweight.
2. Choose the lineweight in the layer palette, usually "Default." This action causes the Lineweight dialog box to appear.

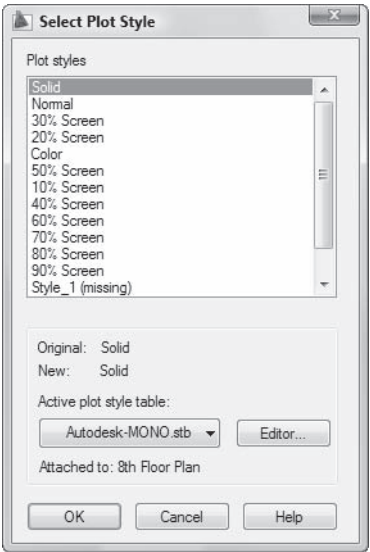


3. Select a lineweight, and then choose **OK**.

Setting the Layer Plot Style

Plot styles determine how all objects residing on a layer are plotted. To change the plot style:

- 1. Select the layer names(s) for which to set a plot style.
- 2. Click the plot style name, typically “Solid.” This causes the Select Plot Style dialog box to appear.



- 3. Choose plot style from those available in the dialog box, and then click **OK**.

Plot styles are available only when the feature is turned on, as described in Chapter 20; otherwise, the plot style names are shown in gray and cannot be changed.

 Setting Layer Print Toggles

You can specify that some layers print, while others do not. Under the Plot column, choose the printer icon for the layer(s) you don’t want to print or plot. To allow the layer to print, simply choose the icon a second time.

Description

The **Description** column permits a descriptive sentence for each layer.

Layer Controls in Layouts

In layout mode (paper space), more columns are added to the layer dialog box. To see them, close the dialog box, and switch to layout mode by selecting any layer tab. Re-open the Layer Properties Manager dialog box, and scroll the layer listing all the way to the right.

Status	Name	On	Freeze	Lock	Color	Linetype	Lineweight	Plot Style	Plot	New VP Freeze	Description	VP Freeze	VP Color	VP Linetype	VP Lineweight	VP Plot Style
	0				white	CONTINU...	Default	Solid					white	CONTINUOUS	Default	Solid
	8th floor f...				200	CONTINU...	0.25 mm	Solid					200	CONTINUOUS	0.25 mm	Solid
	8th floor f...				blue	CONTINU...	0.25 mm	Solid					blue	CONTINUOUS	0.25 mm	Solid

AutoCAD can work with tiled or floating viewports. You can freeze layers in floating viewports — something you cannot do in model view — to display different sets of layers in different viewports.

AutoCAD allows you to do the following to layers in floating viewports: freeze the active viewport; freeze new viewports; and override the color, linetype, lineweight, and plot style in specific viewports.

Layout Freeze Modes

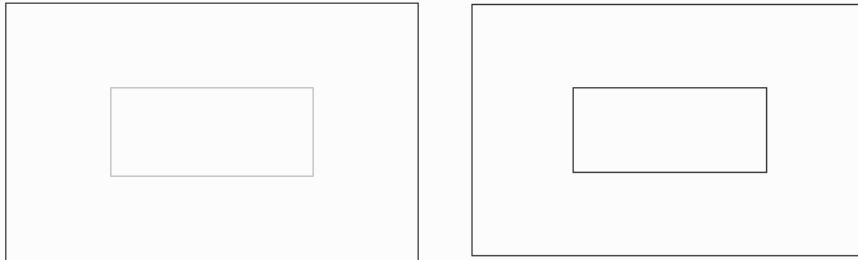
The **New VP Freeze** column freezes specified layers when a new viewport is created. Select the layer name(s), and then choose the icon in the New VP Freeze column. When the icon is a shining sun, the layer is thawed; when the icon is a snowflake, the layer is frozen.

The **VP Freeze** column automatically freezes the specified layers in the current viewport. Select the layer name(s), and then choose the icon in the VP Freeze column (formerly named “Current VP Freeze”). As always, when the icon is a shining sun, the layer is thawed; when the icon is a snowflake, the layer is frozen. When the icon is gray, however, AutoCAD cannot display floating viewports, because the drawing is not in layout mode.

Layout Property Modes

The **VP Color** column allows you to override the layer color for the current viewport. You use it like this:

1. In model space, draw an object, like a rectangle.
2. Switch to a layout tab, and then create two viewports. Double-click a viewport to make it current.
3. Start the **LAYER** command.
4. In the layer dialog box, change **VP Color** from “White” to “Red,” and then exit the dialog box.



Notice that the rectangle is red in one viewport (shown at left, below), but remains black in the other (at right).

The same trick can be played using linetypes, lineweights, and plot styles. Each viewport can display the drawing in a different manner. The viewport is said to have its layer properties “overridden.”

Resetting Overridden Layout Properties

To revert the viewport properties to the layers’ default settings, use the **SETBYLAYER** command. Technically, this command resets the properties of objects to ByLayer. The command applies only to viewports in layouts.

Selecting Layers

Some types of drawings may have many, many layer names. In theory, AutoCAD drawings can have an unlimited number of layers; in practice, drawings may contain thousands. Working your way through long lists of layers can become tedious.

For example, open the *8th floor.dwg* file included on the Companion CD. Start the **LAYER** command, and notice the text on the status line of the dialog box:

All: 225 layers displayed of 225 total layers.

CONTROLLING LAYER FILTERS

Layer filters can slow down AutoCAD, so Autodesk added system variables that help speed up the display of the layers palette and dialog box.

The **SHOWLAYERUSAGE** system variable toggles the display of icons. (The icons indicate whether layers have objects or are empty.) Turn off this system variable (set it to **0**) to improve the speed of the dialog box and palette.

When drawings have more than 99 filters *and* the number of filters exceeds the number of layers, the **LAYERFILTER-ALERT** system variable determines when excess filters can be deleted, thereby improving performance:

LayerFilterAlert	Meaning
0	Filters are never deleted.
1	All filters are deleted without warning the next time the layer dialog box is opened.
2	All filters are optionally deleted; prompts ‘Do you want to delete all layer filters now?’ the next time the Layer dialog box is opened.
3	Filters are selectively deleted; displays dialog box for selecting filters to delete the next time the drawing is opened.

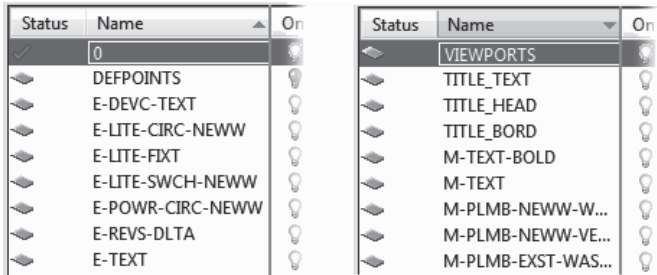
Scroll through the list of layers to appreciate its length. Two hundred and twenty-five is a lot of layers! The reason is that this drawing includes six other drawings, known as “external references.” The original drawing has 27 layers; the xrefs contribute the additional 198 layers.

To help out, AutoCAD allows you to sort and to shorten the list of names using several techniques:

- By sorting names and properties alphabetically in columns.
- By searching for layer names.
- By inverting layer listings.
- By using group and property filters.
- By deleting unused layers.

Sorting by Columns

The names of the columns — **Name**, **On**, **Freeze**, **Lock**, and so on — are actually buttons. For instance, click once on the **Name** column button, and the column sorts layer names in alphabetical order: 0 - 9 followed by A - Z. See figure at left, below.



Click a second time, and the column sorts in reverse order: Z - A followed by 9 - 0. The figure above at right illustrates layer names sorted in reverse order starting with Z.

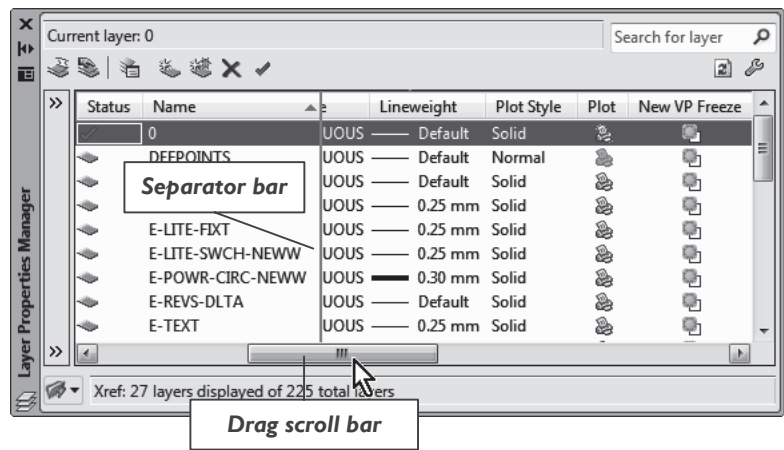
To sort layers by color *number*, click the **Color** column button. The first color is red. Choose it a second time to reverse the order of color numbers, starting with the highest color number used by drawing.

Similarly, clicking the **On**, **Freeze**, and other column titles sorts the layers by state. Click once, and the column sorts by light bulb, sun, or open padlock. Click a second time, and the column reverses the sorting by dim bulb, snowflake, or closed padlock.

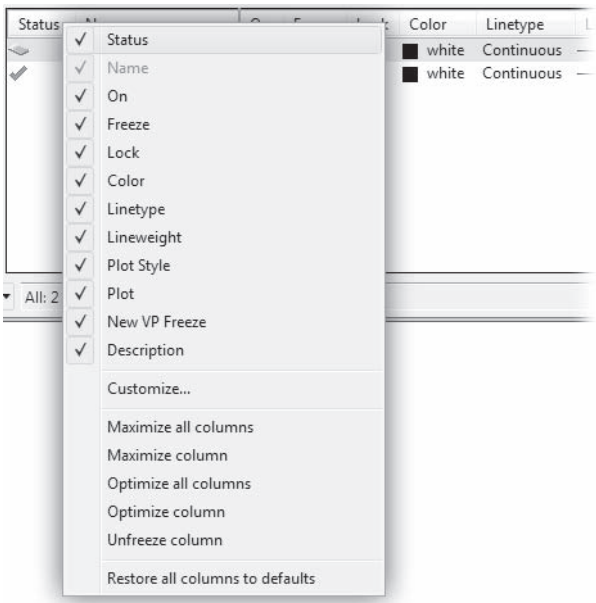


Notes You can change the order of the columns by dragging their headers around. For instance, if you want Color next to On, simply drag the Color header next to On.

The Status and Name columns are “frozen”; as you drag the scroll bar, they remain in view, while other columns slide past.

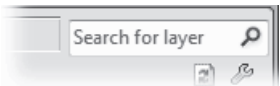


You can hide the display of columns. Right-click any column header, and then select the column name you want hidden. (The checkmarks indicate the headers that are displayed.)



Searching for Layer Names

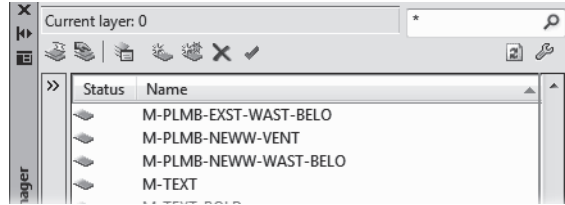
The **Search for Layer** field reduces the list of layer names displayed by the dialog box.



Continuing with the *8th floor.dwg* sample drawing:

- 1. Click in the **Search for Layer** field.

Notice that an asterisk appears (*). This is a “wild card” character. It lists all layer names.



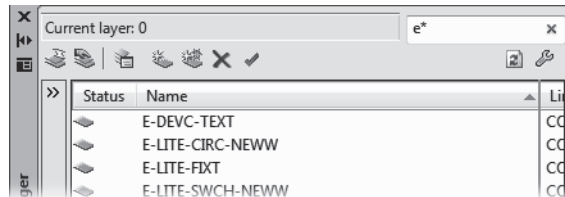
- Let's look for all layers starting with the letters TITLE and ending with any characters. The filter would read:

title*

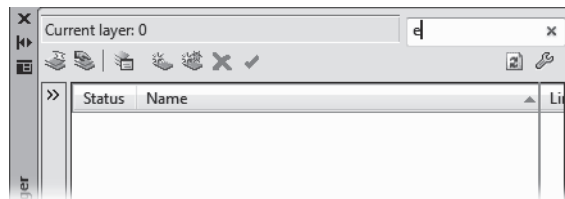
This lists layers named TITLE_BORD, TITLE_HEAD, and TITLE_TEXT. It would not list VIEW_A, VIW2I, or 3RD_VIEW, because the names don't match TITLE*.

Another wild card is the question mark (?), which matches single characters.

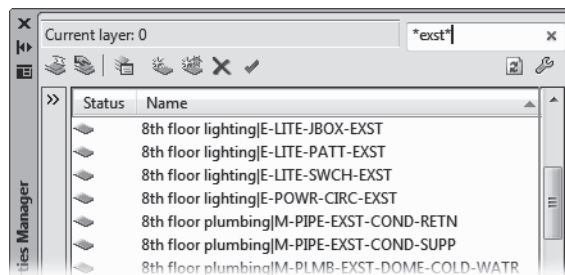
- Enter “e*”. The list shows all layer names that start with the letter E.



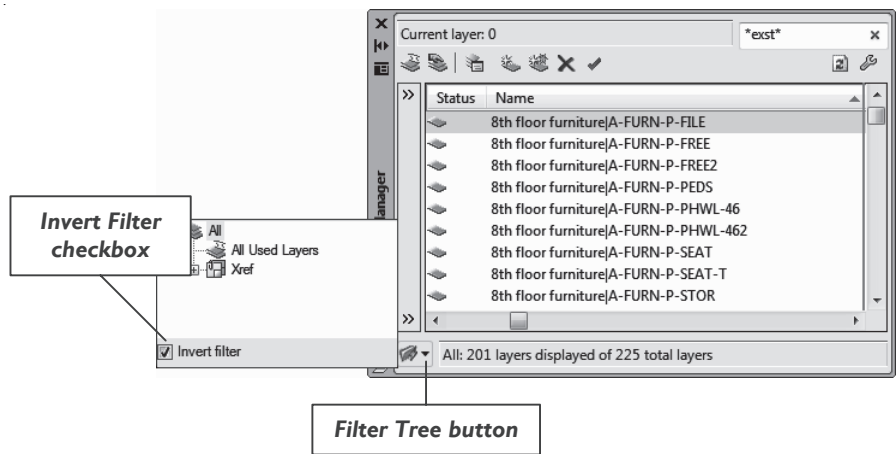
- Now erase the asterisk, leaving the **e** on its own. Notice that no layer names appear, because none consists of just the letter **e**.



- Prefix the search term with * to find all layers with a term. For example, ***exst*** finds all layer names that contain “exst” (short for “existing”):



- Click the **Invert Filter** option to display all layers, except those that contain **exst**:
 - Click the **Filter Tree** button. Notice the flyout.
 - Click the **Insert Filter** checkbox.

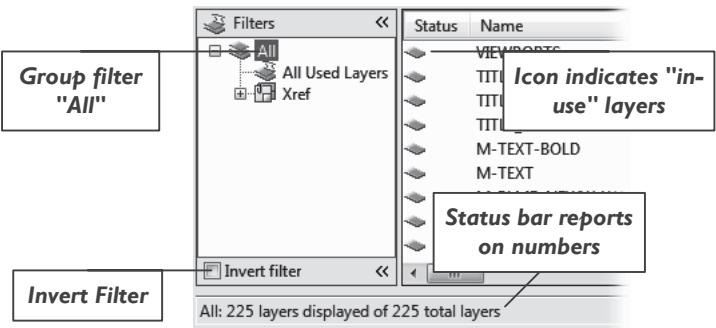


Warning! When the **Invert Filter** is applied to **All**, the layer list is empty.

Creating Group Filters

When drawings have large numbers of layers, it’s useful to organize them into groups and subgroups. For example, you can create a group of all layers to do with electrical or with title block information. AutoCAD includes three groups: All layers, All Used Layers, and Xref (layers from externally-referenced drawings). These work with the Invert Filter option. For example, choose the **Xref** group and then click **Invert Filter**: this forces AutoCAD to display only layers in the base drawing and none from xrefs.

Layer groups are controlled by the *tree view* at the left of the layer palette. It shows groups of layers in a *hierarchy*. The topmost group of the hierarchy is always **All**, and holds the names of all layers. Select it, and the list view (the right half of the dialog box) lists the names of all layers.

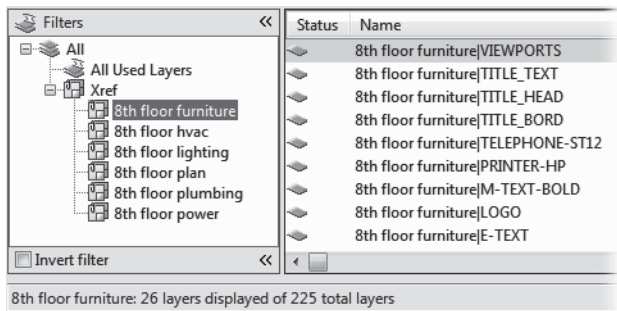


On the status line, AutoCAD reports the number of layers displayed by the filter.

Underneath the All group is the **All Used Layers** group, which contains all layers that have at least one object. When you click it, you see the list of all used layers in the list view. Click **Invert Filter** to see only the names of empty layers, those that contain no objects.

When drawings contains xrefs (short for “externally referenced drawings”), the tree view also includes a group named **Xref**. Click it to see the names of all layers found in xrefs. Below the **Xref** group are the names of the xref drawings. Each is a subgroup that contains the names of layers found in that particular drawing.

Notice that the status line now reports that just 26 layer names are displayed.



Group filters are groups of layers that you drag from the list view into the group filter name. Group filters are static; they change only when you add and subtract layers to and from the group.



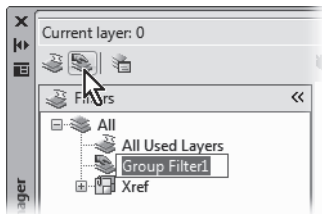
Notes To view all layers *not* in xrefs, select the **Xref** group, and then turn on the **Invert Filter** option.

Although layers may appear to be empty, sometimes they contain objects from block definitions.

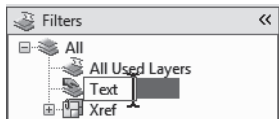
TUTORIAL: CREATING GROUP FILTERS

In this tutorial, you create a group filter that lists all text-related layers. Follow these steps:

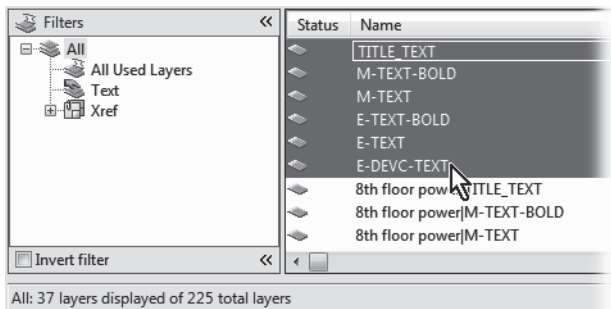
- 1. Start AutoCAD with the *8th floor.dwg* drawing, found on the Companion CD.
- 2. Enter the **LAYER** command to open the Layer Properties Manager dialog box.
- 3.  Click the **New Group Filter** button. (Alternatively, press **ALT+G**.)
Notice that AutoCAD creates an empty filter with the generic name of “Group Filter 1.”



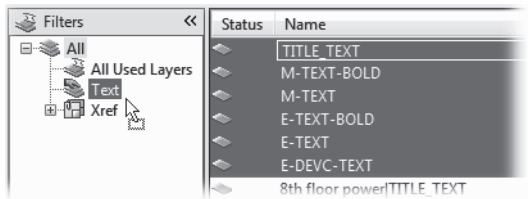
- 4. Change the name to something more meaningful: **Text**.



- 5. Now it's time to populate the group filter with all layer names that contain “text”:
 - a. Click the **All** group filter to display all layer names.
 - b. In the search bar, enter “*text*” to select all layer names containing “text.”
 - c. Select the layer names. (Choose the first name, hold down **SHIFT**, and then choose the last one.)

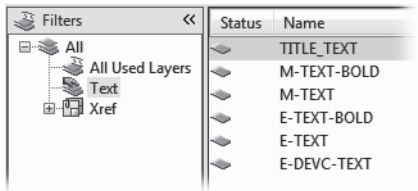


6. Drag the selected name over to the Filters pane, and deposit them on the **Text** name.



Don't worry! When you drag layer names around this dialog box, you do not move or lose them; AutoCAD copies them.

7. Select the **Text** filter. Notice that the list view shows a shortened list of layer names.



Select the **All** filter to see all names again.

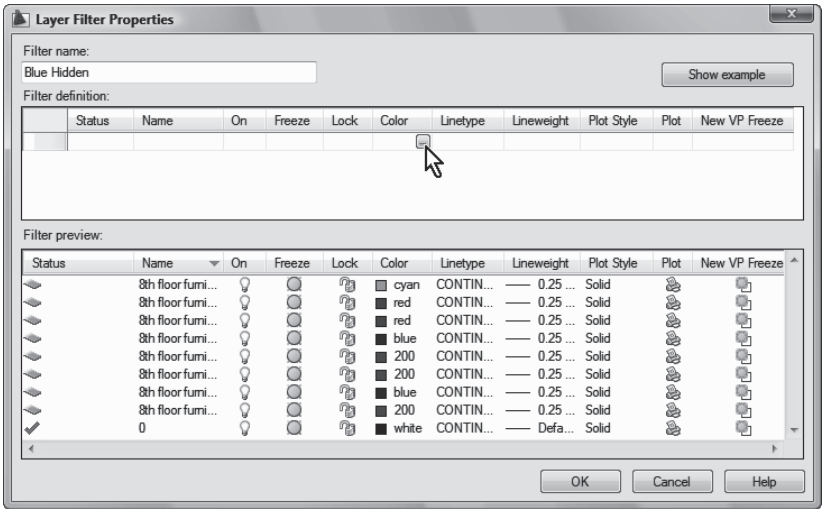
TUTORIAL: CREATING PROPERTY FILTERS

AutoCAD also supports *property* filters. These filters reduce the number of layer names based on their properties — color, linetype, frozen status, and so on. For example, you could have a group that holds layers with blue color and Hidden linetype. (In contrast, group filters are based on layer names only.)

Property filters are dynamic; as you create (or delete) layers of the same properties, AutoCAD automatically adds and removes them from the filter group. There is no need to drag layer names into property filters, as you had to with group filters.

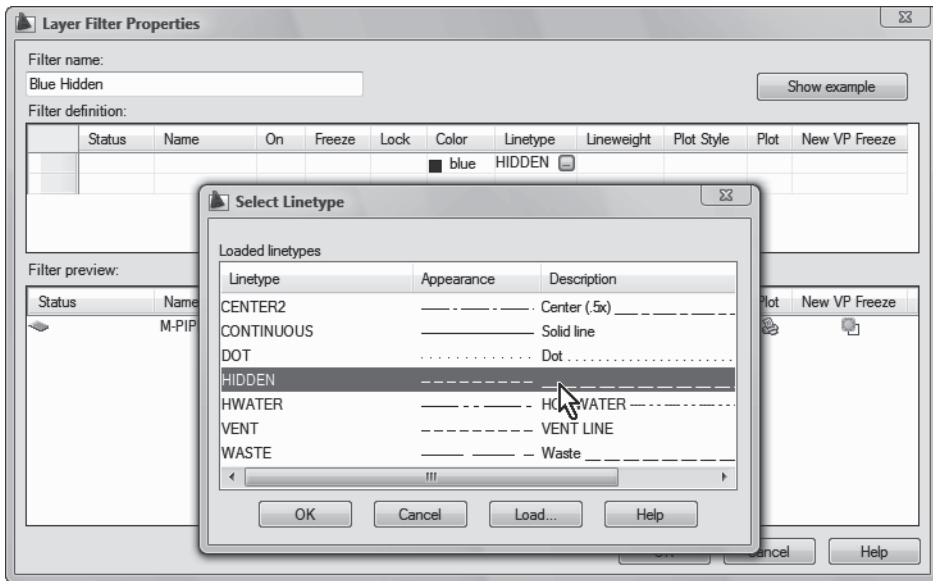
In this tutorial, you create a group filter that lists all layers that are colored blue and have the Hidden linetype. Follow these steps.

- 1. Start AutoCAD with the *8th floor.dwg* drawing, found on the Companion CD.
- 2. Enter the **LAYER** command. Notice the Layer Properties Manager palette.
- 3.  Click the **New Property Filter** button. (Alternatively, press **ALT+P**.)

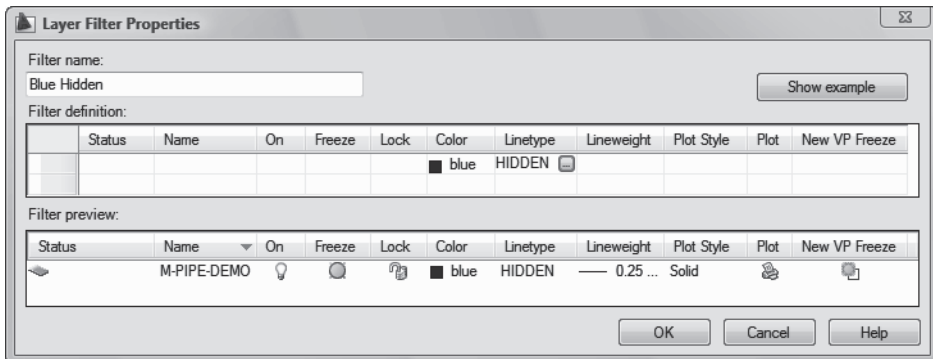


Notice that AutoCAD displays the Layer Properties Filter dialog box. In the upper half, you define the filter(s); the lower half displays the names of layers and their properties.

4. In the **Filter Name** field, replace the generic name “Properties Filter 1” with:
Filter Name: **Blue Hidden**
5. Under each heading (Status, Name, On, ...) you can enter a restriction that shortens the list. In this tutorial, this filter is supposed to display just those layers colored blue:
 - a. Under **Color**, click the blank area. Notice the small button that appears.
 - b. Click the **...** button. Notice the Select Color dialog box.
 - c. Select the blue color (index color 5), and then click **OK**.
6. Apply a further restriction: just those layers with the Hidden linetype. Repeat the steps of #4, but this time select the **Hidden** linetype when the Select Linetype dialog box appears. In the lower half of the dialog box, the list of layers is again shortened to blue layers with Hidden linetype.
7. Click **OK**.
8. Select the **Blue Hidden** property filter, and only one layer appears.
9. Create a new layer with color blue and linetype Hidden. Notice that it is automatically added to the property filter.



In the lower half of the dialog box, notice that the list of layers is shortened: just the blue layers are listed.



10. Click **OK** to exit the Layer Filter Properties dialog box.
11. Back in the layer palette, notice that the Blue Hidden filter takes its place in the Filters tree. Click it to see the names of layers with blue and hidden linetype properties.



Notes To edit a property filter, double-click its name in the tree view. This causes the Layer Properties Filter dialog box to appear.

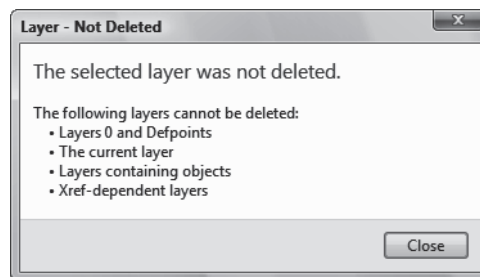
Group and Property filters apply only to the names of layers appearing in the layer palette. They have no effect on the drawing.

Deleting Layers

AutoCAD allows you to delete *empty* layers, those with no objects.

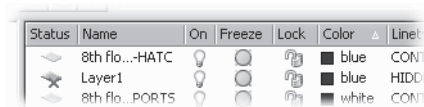
But you cannot delete layers 0 and DefPoints, as well as externally-referenced layers — even when these are empty. Sometimes, empty layers cannot be erased, because they are part of an unused block definition.

Select one or more layer names, and then choose the **Delete Layer** button. (Alternatively, press ALT+D.) If the layer cannot be erased, AutoCAD displays the following dialog box to explain why not:



If the layer is empty, then it is erased immediately.

(In the layer dialog box, the layer is not erased right away. Rather, an icon in the Status column marks the layer for deletion; this allows you to change your mind. The layer is deleted when you click **Apply** or **OK**.)



Layer States Manager

The **Layer States Manager** saves the current state of layers, and then restores the state at a later time. The *state* of layers includes names and properties, such as whether they are thawed or frozen, as well as their colors and linetypes.

Once you save a layer state by name, you can edit the state, and export it for sharing with others.

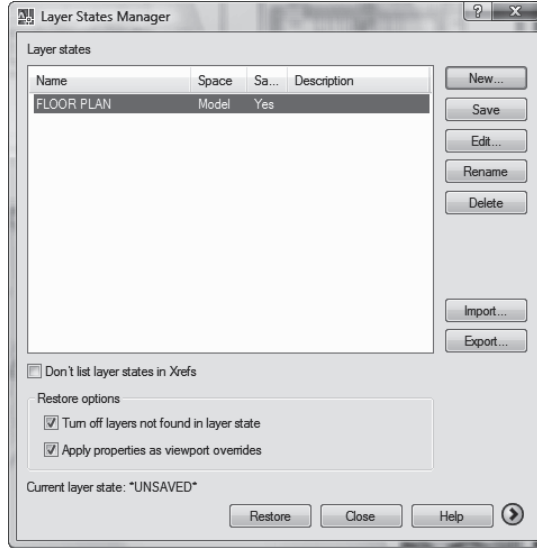
TUTORIAL: CREATING AND APPLYING LAYER STATES

1. To create layer states, start the **LAYERSTATES** command with one of these methods:
 - In the ribbon's 2D Home tab, click the **Layer States Manager** button on the Layers panel.
 - Or, at the 'Command:' prompt, enter the **layerstates** command:

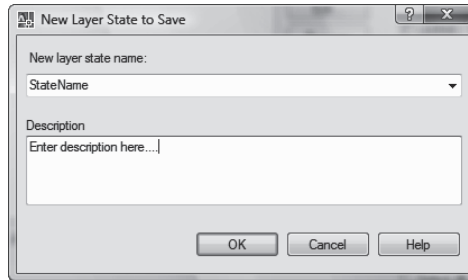
Command: **layerstates** (Press ENTER.)

- Alternatively, enter the aliases **la** or **ddlmodes** (an old name for this command) at the 'Command:' prompt.
- You can also access the dialog box through the Layer Properties Manager palette.

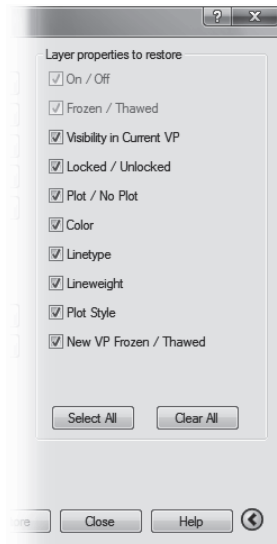
In all cases, AutoCAD displays the Layer States Manager dialog box.



2. To create a new layer state, click **New**. Enter a name and description, and then click **OK**.

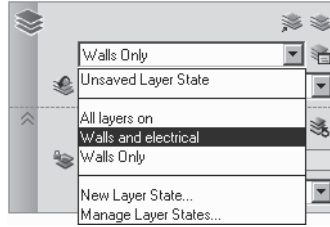


3. Now you can determine which properties will be saved. Click the **More** button to expand the dialog box, and then choose from the list of properties.



4. To save the layer state to disk, click the **Export** button. Provide a name and folder for the .lay file, and then click **Save**.

- The easy way to apply a layer state is to select its name from the ribbon's Layers panel.



To import a layer state, follow these steps:

- Click the **Import** button.
- Select the folder and name of a file, and then click **Open**.

Duplicate layer states names are not imported. As of AutoCAD 2008, you can import layer states from *.dwg* (drawing), *.dws* (drawing standards), and *.dwt* (drawing template) files — as well as from *.las* (layer state) files.

CONTROLLING LAYERS: ADDITIONAL METHODS

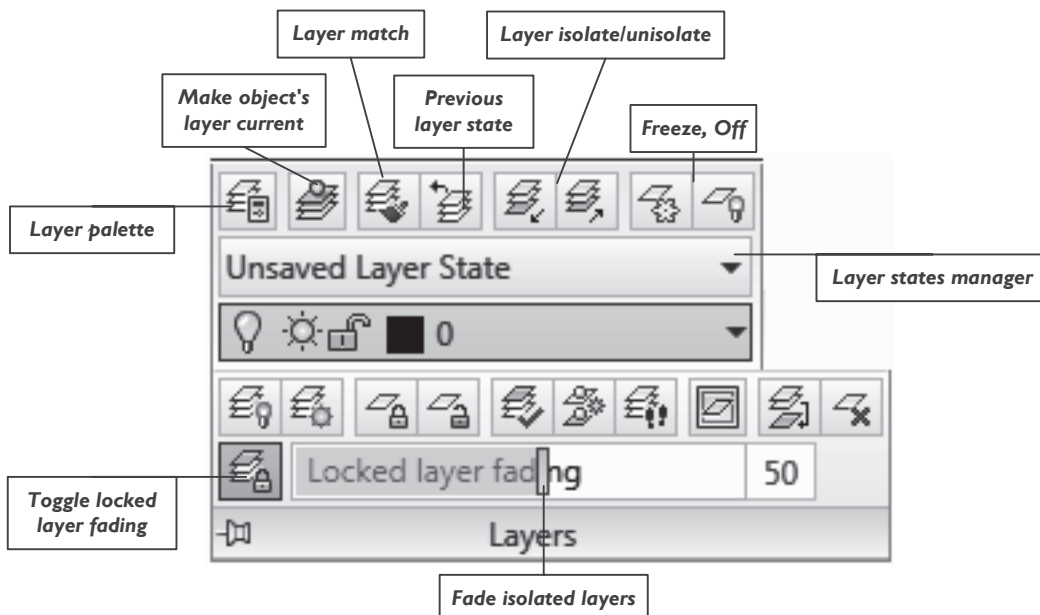
AutoCAD provides several other methods to control layers:

- Layers** panel controls layers on the ribbon .
- CLAYER** system variable makes a layer current quickly.
- LAYMCUR** command makes the selected object's layer current (formerly **AI_MOLC**).
- LAYERP** command restores the previous layer state.
- Layer Tools** manipulate layers through individual commands.
- LAYISO** command isolates selected layers by freezing or locking all other layers.

Let's look at each of these.

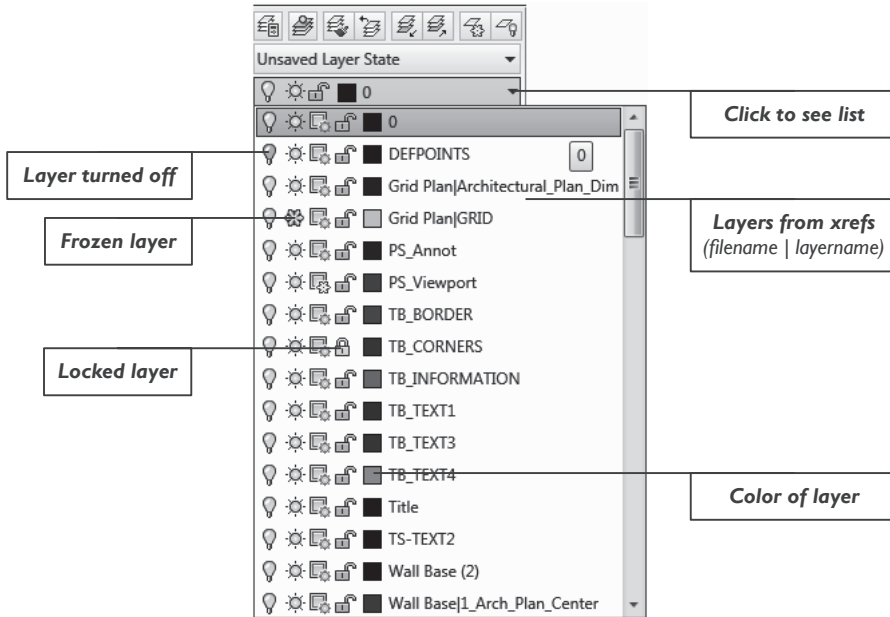
Layers on the Ribbon

The ribbon's Layer panel select the current layer without using the **LAYER** command. (The layout of this panel changed in AutoCAD 2010.)



Changing the Current Layer

To change the current layer, click the down arrow shown below. It reveals a list of layer names, along with icons signifying their status. (The actual names of layers vary according to the drawing currently open.)



Select the name of a layer to make it current. It's that simple.

There is, unfortunately, one catch. You cannot make current any layer whose name is shown with a snowflake, because it is frozen. (You cannot see or work with frozen layers).

Changing the Layer's Status

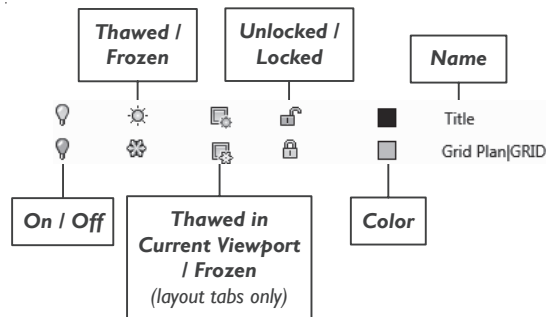
The drop-down box lists a quintet of icons beside each layer name. The colored square icon represents the color assigned to this layer. Each of the other four icons has states:

Lightbulb on or off — layer is on (default) or off.

Sun or snowflake — layer is thawed (default) or frozen.

Sun or snowflake on square — layer is thawed (default) or frozen in the current viewport; this icon changes only when the drawing is in paper space (layout mode).

Padlock — open or closed: layer is unlocked (default) or locked.



You change the status of each of the four icons simply by clicking it. Note that the thawed/frozen in current viewport icon changes only when the drawing is in layout mode (paper space). Clicking the color square displays the Select Color dialog box, allowing you to change the layer color ([new to AutoCAD 2010](#)).



LayMCur

An easy way to switch to another layer is with the LAYMCUR command (short for “Layer Make Current”; formerly the AI_MOLC command). Suppose you are interested in switching to the layer holding a green dotted line whose name you’re not sure about. You use the feature as follows:

1. Choose the **Make Object’s Layer Current** button. AutoCAD prompts you:
Select object whose layer will become current:
2. Select the green dotted line. AutoCAD reports:
layername is now the current layer.

Check the toolbar, and you will see that layer *layername* is now current.

CLayer

If you prefer the keyboard to dialog boxes, then CLAYER system variable (short for “current layer”) is the fastest way to switch between layers, as follows:

Command: **clayer**

New value for CLAYER <default>: *(Enter the name of the layer, and then press ENTER.)*

AutoCAD immediately makes that layer current.



LayerP and LayerPMode

The LAYERP command (short for “layer previous”) undoes changes to layer settings, much like the ZOOM Previous command restores the previous view.

Command: **layerp**

Restored previous layer status.

The command, however, cannot undo the changes to renamed, deleted, purged, and newly-created layers. (*Purging* is described in the next chapter.) When you rename a layer and change its properties, only the properties are changed back, not the name.

A related command is LAYERPMODE (short for “layer previous mode”). It toggles whether AutoCAD tracks changes to layers.

Command: **layerpmode**

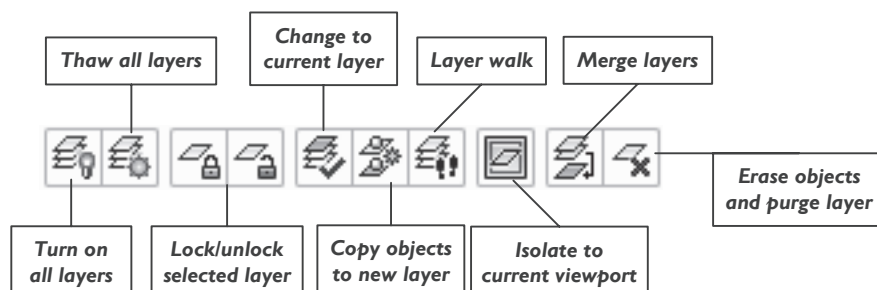
Enter LAYERP mode <ON>: *(Enter ON or OFF.)*

When turned off, the LAYERP command does not work, and so AutoCAD reminds you:

Layer-Previous is disabled. Use LAYERPMODE to turn it on.

Layer Tools

The layer tools were formerly part of Express Tools, a collection of commands not supported by Autodesk. In AutoCAD 2007, Autodesk changed the status of layer tools to fully-supported. Many of them now appear as buttons on the ribbon’s Layer panel.



The layer tools are:

LAYCUR changes the layer of selected objects to that of the current layer.

LAYMCH changes the layers of selected objects to that of a selected object.

LAYMCUR makes the selected object's layer current, just like the **AI_MOLC** command.

LAYMRG moves objects to another layer, and then removes the layer of the moved objects.

LAYWALK displays objects on selected layers.

LAYVPI isolates the selected object's layer in the current viewport by freezing its layer in all other viewports. This command works only in paper space with two or more viewports.

LAYISO turns off all layers except those holding selected objects (detailed below). **LAYUNISO** turns on layers that were turned off with the last **LAYISO** command.

LAYDEL erases all objects from the specified layer, and then purges the layer from the drawing. The current layer cannot be deleted. If you make a mistake, you can use the **U** command later to restore the layer name and its objects.

LAYFRZ freezes the layers of the selected objects. **LAYTHW** thaws all layers.

LAYLCK locks the layer of the selected object. **LAYULK** unlocks the layer of a selected object.

LAYOFF turns off the layer of the selected object. **LAYON** turns on all layers, except frozen layers.



LAYISO

The **LAYISO** command (short for “layer isolation”) isolates one or more layers, causing the objects on all other layers to fade away — hence isolating the selected ones. The other layers are either frozen (made invisible) or faded — your choices. When faded, layers are locked, not frozen; this lets you see but not edit them.

The **LAYLOCKFADECTL** system variable determines the amount of fading. You may find it easier to use the Locked Layer Fading slider on the ribbon's Layer panel. The maximum fade amount is 90% (not quite entirely faded); the default is 50%.

The related **LAYUNISO** command returns layers to normal.

Let's take a look at how this works.

1. From the Companion CD, open *property.dwg*.
2. Start the **LAYISO** command:
 - In the ribbon's 2D Home tab, click the **Layer Isolate** button in the Layers panel.
 - At the 'Command:' prompt, enter the **layiso** command:

Command: **layiso**

Current setting: Lock layers, Fade=50
3. Type “s” to specify the Settings options:

Select objects on the layer(s) to be isolated or [Settings]: **s**

Enter setting for layers not isolated [Off/Lock and fade] <Lock and fade>:

The Settings options have the following meaning:

- **Off** freezes layers.
- **Lock and fade** locks and fades layers:

- **Fade value** specifies the percentage to fade locked layers, from 0% to 90%. I find that the maximum of 90 works most effectively.

Enter fade value (0-90) <50>: **90**

4. Select one or more objects on the layers you want to *isolate* (display fully, not fade). For this tutorial, select one of the black property lines; this will fade the blue text and the dimensions.

Select objects on the layer(s) to be isolated or [Settings]: *(Pick a black property line.)*

Picking one object selects its layer, and hence automatically selects all other objects on the layer.

5. Press **ENTER** to exit object selection.

Select objects on the layer(s) to be isolated or [Settings]: *(Press ENTER.)*

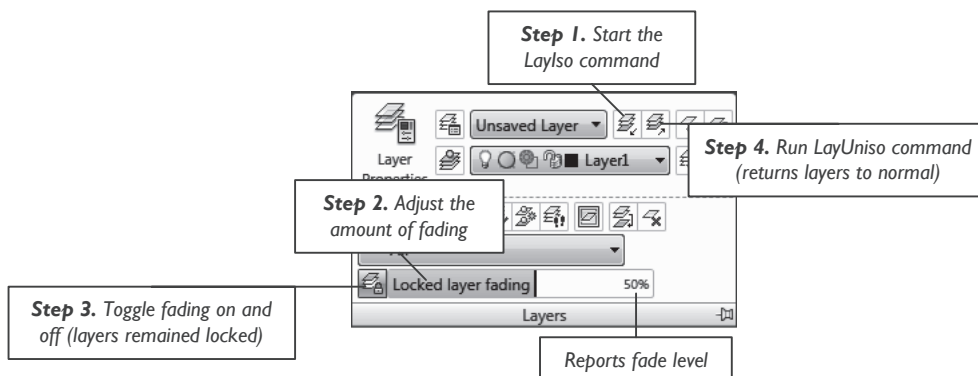
Layer Property Lines has been isolated.

AutoCAD reports the name of the layer(s) isolated. Notice how the objects on the other (unselected) layers are isolated (faded), as illustrated below:



You may find it much easier to use the ribbon's Layers panel to control layer isolation; for example, the Locked Layer Fading slider lets you interactively control the amount of fading displayed by locked layers.

Follow the four steps illustrated below to use the panel's layer isolation controls.



EXERCISES

1. In this exercise, you change the colors of objects.
 - a. Open AutoCAD to start a new drawing.
 - b. Use the **LAYER** command to set the color of layer "0" to cyan (light blue).
 - c. Draw some lines. Are they drawn in color?
 - d. Use the **COLOR** command to set the current color to yellow.
 - e. Draw three circles. Did the **COLOR** command override the layer's color setting?
 - f. Next, set the current color to red.
 - g. Draw three boxes. Do they appear in the correct color?
 - h. Start the **PROPERTIES** command, and then select all the objects on the screen.
 - i. Select blue as the color. Did all the objects change to blue?

2. In this exercise, you create a layer, and then freeze and thaw it.
 - a. Create a layer named "Mylayer."
 - b. Set "Mylayer" as the current layer. Do you see the layer name in the ribbon at the top of the screen?
 - c. Set the layer color to green, and then draw some objects on the layer.
 - d. Using the Layer droplist, freeze layer "Mylayer", and then set the current layer to "0." Do the objects you drew disappear?
 - e. Thaw "Mylayer". Do the objects reappear?

3. In this exercise, you create several layers, and then change their properties.

With the **LAYER** command, create seven layers using these names:

 - Landscape
 - Roadways
 - Hydro
 - ToBeRemoved
 - StormSewer
 - CableTV
 - Building

Using the layer palette:

 - a. Change the color of layer "Landscape" from white to green.
 - b. Change the lineweight of layer "Roadways" from Default to 0.083" (2.11 mm).
 - c. Change the linetype of layer "Hydro" from Continuous to Gas_line.
 - d. Change the "ToBeRemoved" layer from white to red, and from Continuous to Dashed linetype.
 - e. Change the linetype of the "CableTV" layer from Continuous to Hidden, and change the lineweight to 0.020" (0.51 mm).
 - f. Make the "CableTV" layer current by choosing the **Current** button.
 - g. Draw some lines. Do they appear in hidden linetype? If the lines do not look thick, ensure the **LWT** button is depressed on the status line.
 - h. Use the Layer droplist to change to the "ToBeRemoved" layer, and then draw some lines. Do they appear red and dashed?
 - i. Again, use the Layer droplist to change to the "Building" layer, and then draw some lines. Do they appear black and continuous?

4. In this exercise, you copy the properties from one object to all others in the drawing. Continue with the drawing you created in the previous exercise.

Start the **MATCHPROP** command.

Select source object: *(Select one of the red, dashed lines drawn on the “CableTV” layer.)*

Select destination object(s) or [Settings]: **all**

Select destination object(s) or [Settings]: *(Press ENTER.)*

Do all the objects become red and dashed?

CHAPTER REVIEW

1. What do hidden lines show in drawings?
2. Name two ways linetypes can be applied to objects in AutoCAD:
 - a.
 - b.
3. How do you control the length of individual segments in dashed lines?
4. How do you load all linetypes into a drawing?
5. What is the difference between *global* and *local* linetype scaling?
6. Is it possible to save the current state of layers?

If so, how?
7. Describe five properties that the **PROPERTIES** command changes.
8. What is meant when the color of objects is changed to **ByLayer**?
9. Why would you use the **PROPERTIES** command instead of the Properties panel on the ribbon?
10. Can the Properties panel change objects?

If so, how?
11. What is the **MATCHPROP** command used for?
12. Why would a CAD drafter use layers?
13. What layer option do you use to create new layers?
14. How do you turn on frozen layers?
15. What is the difference between *locking* layers and *freezing* layers?
16. How do you obtain a listing of all layers in drawings?
17. Can you have objects of more than one color on the same layer?

Explain.
18. What do the following layer symbols mean?
 - Snowflake
 - Open lock
 - Lightbulb glowing
 - Printer
 - Colored square
19. Name a benefit to using lineweights.
20. Is it possible to add custom linetypes to drawings?

Custom lineweights?
21. What color is designated by R?

What is color 7?

What is a *color book*?
22. Describe what is meant by *RGB* in terms of colors.
23. Should you use lineweights to represent objects with width?
24. What is the purpose of the **LWT** button on the status bar?
25. Describe the steps to changing the lineweight of circles using the Properties palette:
 - a.
 - b.
26. Explain how these three classes of linetype differ:
 - Simple
 - ISO
 - Complex

27. Before using linetypes in new drawings, what must you do first?
28. What is the name of the linetype used to show hidden edges?
29. **LTSCALE** is set to 0.5 and **CELTSCALE** is set to 5.0. At what scale is the next linetype drawn?
30. Under what condition can layers be deleted?
Can layer 0 be deleted?
31. Describe two ways to manage long lists of layer names:
 - a.
 - b.
32. When drawings are in model space, why do the **Current VP Freeze** and **New VP Freeze** columns not appear in the Layer palette?
33. What is the purpose of the **Show all used layers** filter in the layers palette?
34. Can you assign lineweights to layers?
Assign linetypes?
Assign hatch patterns?
35. When linetypes are assigned to layers, are all objects on those layers displayed with that linetype?
If not, why not?
36. Explain the purpose of the following filters:
Group
Properties
37. What is the purpose of the **CLAYER** system variable?
38. Which keystroke shortcut displays the Properties palette?
39. Describe how the **PICKADD** system variable affects object selection.
40. Can the Properties palette be used to change the endpoints of line segments?

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Constructing Objects

Drawing the same objects over and over was a tedious part of hand drafting. In Chapter 6, “Drawing with Efficiency,” you learned to insert blocks to place parts quickly in drawings. In some circumstances, however, blocks are not the best method.

This chapter introduces other means of making copies — mirrored copies, parallel offsets, and arrays of copies. After completing this chapter, you will be able to use the following AutoCAD’s commands for constructing objects from existing objects:

COPY makes one or more identical copies of objects.

MIRROR makes mirrored copies.

MIRRTEXT determines whether text is mirrored.

OFFSET makes parallel copies.

MEASURE and **DIVIDE** place copies of points and blocks along objects.

ARRAY constructs linear, rectangular, and polar copies.

FILLET and **CHAMFER** create rounded and angled corners.

JOIN joins similar objects into one.

REVERSE reverses the direction of objects ([new to AutoCAD 2010](#)).

REVCLOUD creates revision clouds.

MARKUP inserts marked-up .dwf files from Design Review.

NEW TO AUTOCAD 2010 IN THIS CHAPTER

- **REVERSE** reverses the direction of lines, polylines, arcs, and so on.
- **JOIN** now joins splines and helixes.



The **COPY** command makes one or more copies of objects.

The copies are identical to the original; all that changes is their location in the drawing. This command is meant for use within drawings; to copy objects between drawings, use the **COPYCLIP** command.

To use the **COPY** command, you need to tell AutoCAD three things: (1) the objects to be copied, (2) the point from which the copying takes place, and (3) the location to place the copies. The most recent displacement is the default value the next time the **COPY** command is used — but only if the **Displacement** option is used. The **COPY** command repeats until you press **ESC**.

TUTORIAL: MAKING COPIES

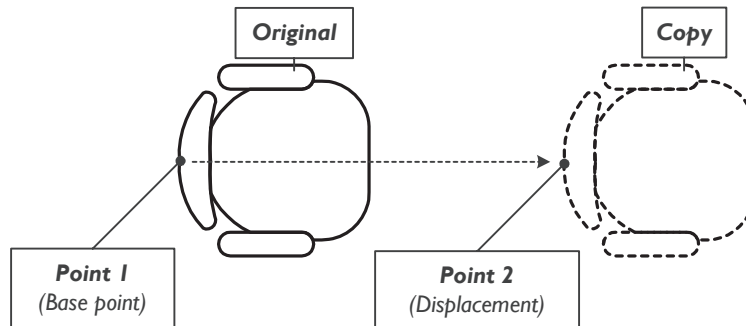
1. To copy one or more objects, start the **COPY** command:
 - In the ribbons's 2D Home tab, choose the **Copy** button from the Modify panel.
 - At the 'Command:' prompt, enter the **copy** command:

Command: **copy** (Press ENTER.)
 - Alternatively, enter the aliases **co** or **cp** at the 'Command:' prompt.
 - You can also use **CTRL+C** and **CTRL+V** to copy and paste objects in the current drawing, to other drawings, and to documents in other applications.
2. In all cases, AutoCAD prompts you to select the objects you want copied:

Select objects: (Pick one or more objects.)

Select objects: (Press ENTER to end object selection.)
3. Identify the point from which the displacement is measured:

Specify base point or [Displacement] <Displacement>: (Pick point 1.)



4. Identify the location for the copied object:

Specify second point of displacement or <use first point as displacement>: (Pick point 2.)
5. The command repeats the last prompt so that you can place additional copies. Pick additional points, or press **ESC** to end the command.

Specify second point or [Exit/Undo] <Exit>: (Press ESC.)

Students often have difficulty understanding the concept of *displacement*. This is the distance from the original object (known as the *base point*) to the location of the copied object. (See the detailed discussion below.)

You can choose any point you like as the base point, but some points make more sense than others. Examples include the lower-right corner of rectangles, the center of circles, and the 0,0 origin of

drawings. In addition, it is often handy to use object snaps, such as INTERsection or INSertion, to pick base points precisely.

To copy objects vertically or horizontally, hold down the **SHIFT** key to turn on ortho mode. (If ortho mode is already on, **SHIFT** turns it off temporarily.)

AutoCAD has several ways to determine the displacement, which you learn about in the next tutorial.

MAKING COPIES: ADDITIONAL METHODS

This command also has the following options:

- **Displacement** specifies the displacement distance.
- **COPYMODE** system variable determines whether the command repeats.

Displacement

The **Displacement** option displays the displacement from the prior use of the **COPY** command:

Specify base point or [Displacement] <Displacement>: **d**

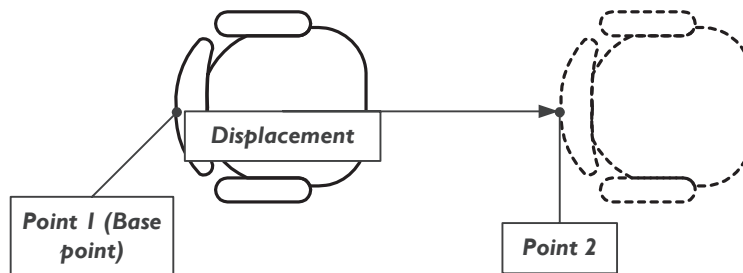
Specify displacement <1.0000, 2.0000, 0.0000>: (Press **ENTER**, or enter a new displacement.)

Whether you press **ENTER** or enter coordinates, AutoCAD displaces the selected objects, and then exits the **COPY** command.

More on Displacements

When AutoCAD prompts, ‘Specify base point or [Displacement]:’ this is a hint of what is to come.

Specify base point means you can enter x, y coordinates (such as 4,5) or pick a point in the drawing. Both methods can be used as the base point, but the x, y coordinates could be interpreted differently by AutoCAD. *Displacement* is a hint that the prompt to follow will select one of the two methods, depending on your next action.



Specify second point of displacement means that you should enter another x, y coordinate or pick another point. Both actions place the copy at a distance that AutoCAD calculates from the two sets of x, y coordinates or two pick points. You can, of course, mix and match coordinate entry and pick points.

It's not clear from the prompt, but AutoCAD wants you to press **ENTER** at <use first point as displacement>. AutoCAD interprets the coordinates you entered at the earlier prompt (such as 4,5) as relative distances. In this case, there is no need to use the **@** prefix. AutoCAD places the copy 4 units right and 5 units up from the original.

This option can, unfortunately, have unexpected results, because, as this book's technical editor Bill Fane once remarked, "Pressing **ENTER** sometimes makes the copied objects end up near Hawaii — not a problem," he adds, "if you live in Maui." This problem occurs when you use the mouse to pick the first point, and then just press **ENTER** for the second point; AutoCAD interprets the second

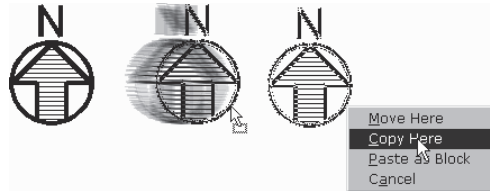
point as 0,0. He recommends using the **ZOOM Extents** command to find the “missing” copy.

1. Start the **COPY** command, and then select objects to copy:
 Command: **copy**
 Select objects: *(Select one or more objects.)*
 Select objects: *(Press ENTER.)*
2. Enter coordinates for the base point:
 Specify base point or [Displacement] <Displacement>: **4,5**
3. Press **ENTER** to place the copy by a relative distance and end the command:
 Specify second point of displacement or <use first point as displacement>: *(Press ENTER to interpret the first point as relative coordinates.)*
 Specify second point or [Exit/Undo] <Exit>: *(Press ENTER to exit the command.)*



Notes You may enter x,y coordinates for 2D displacement, or x,y,z coordinates for 3D displacement. In addition, you can use direct distance entry to specify the displacement.

As an alternative to the **COPY** command, you can use the right-click copy/move/insert technique.



Or, use the **Copy** option of grips editing:

**** STRETCH ****

Specify stretch point or [Base point/Copy/Undo/eXit]: **c**

Become familiar with AutoCAD's many object selection modes, because they are crucial for working efficiently with the commands in this (and the next) chapter. You may wish to review the **SELECT** command in Chapter 6, “Drawing with Efficiency.”

MIRROR

The **MIRROR** command makes copies that are mirrored.

The command saves time when you are drawing symmetrical objects. Draw a half, or a quarter, of a group of objects, and then construct the others by mirroring them. You have the option to retain or delete the originals, as well as to decide whether text should be mirrored or not.

Other drawing programs use the phrases “flip horizontal” and “flip vertical” in place of mirror. As alternatives to this command, you can use the **Mirror** option found in grips editing, or insert blocks with a negative x- or y-scale factor (which mirrors the block).

To use the **MIRROR** command, you need to tell AutoCAD three things: (1) the objects to be mirrored, (2) the line about which the mirroring takes place, and (3) whether the source objects should be erased.

TUTORIAL: MAKING MIRRORED COPIES

- To make mirrored copies of one or more objects, start the **MIRROR** command:
 - In the ribbon's 2D Home tab, choose the **Mirror** button from the Modify panel.
 - Or, at the 'Command:' prompt, enter the **mirror** command:

Command: **mirror** (Press ENTER.)

- Alternatively, enter the **mi** alias at the 'Command:' prompt.
- In all cases, AutoCAD prompts you to select the objects you want mirrored:

Select objects: (Pick one or more objects.)

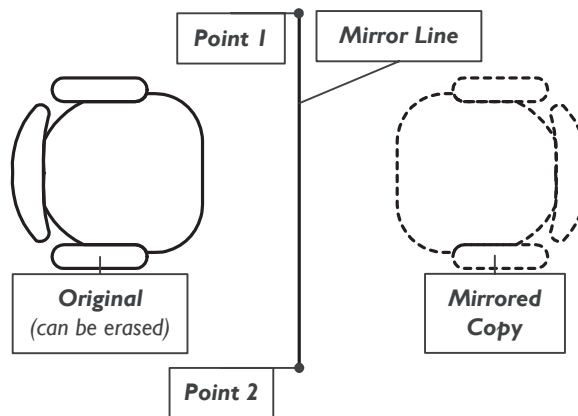
Select objects: (Press ENTER to end object selection.)

- Identify the points that define the mirror "line":

Specify first point of mirror line: (Pick point 1.)

Specify second point of mirror line: (Pick point 2.)

The *mirror line* is the line about which the objects are mirrored. It need not be an actual line; two points will do.



- Decide whether you want the original object(s) erased:

Delete source objects? [Yes/No] <N>: (Enter Y or N.)



Notes To make the mirror line absolutely horizontal or vertical, hold down the **SHIFT** key to turn on ortho mode temporarily.

Polar mode is handy for making mirrored copies at specific angles.

MAKING MIRRORED COPIES: ADDITIONAL METHOD

Whether text is mirrored is determined by the **MIRRORTTEXT** system variable.



Sometimes objects contain text. The dilemma is whether to mirror the text, which makes it read backwards — or not mirror, which makes it read normally.

The MIRRTEXT system variable determines this, as follows:

Command: **mirrtext**
New value for MIRRTEXT <0>: (Enter **1** or **0**.)

The 0 and 1 have the following meaning:

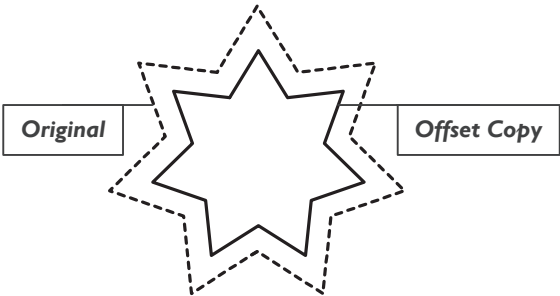
MirrorText	Comment
0	Text is not mirrored.
1	Text is mirrored (default in AutoCAD 2004 and earlier).

Entering **0** produces non-mirrored text, which is the default setting. Enter **1** if you need the text to be mirrored.

 OFFSET

The OFFSET command makes parallel copies.

This command constructs copies that are parallel to the source object; AutoCAD limits you to making offset copies of one object at a time. When the objects have curves or are closed, the copies become larger or smaller, depending on whether they are on the inside or outside of the source.



To create offset copies, AutoCAD needs to know three pieces of information, and in this order: (1) the offset distance, (2) the objects to offset, and (3) the side on which to place the offset copies.

It may seem counter-intuitive *first* to specify the distance and *then* to select the object, but “that’s the way the Mercedes bends,” as the driver said after his automobile accident. (Pun credit: technical editor.)

AutoCAD offsets lines, arcs, circles, ellipses, elliptical arcs, polylines (2D only), splines, rays, and xlines. Sometimes, unexpected results happen, as illustrated below. The dashed lines are copied offset from the original polyline (shown as the heavy line).

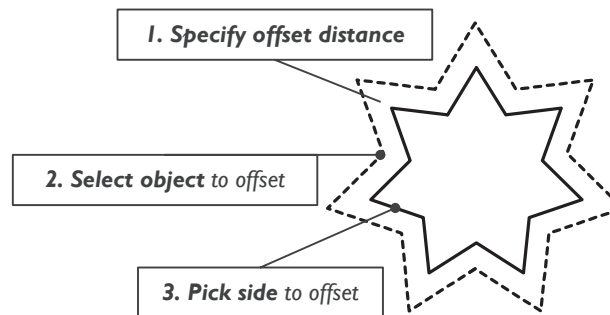


The thick line indicating the original polyline; the dashed lines, the offset copies.

As an alternative to this command, use the **MLINE** command to place parallel lines.

TUTORIAL: MAKING OFFSET COPIES

1. To make offset copies of an object, start the **OFFSET** command:
 - In the ribbon's 2D Home tab, choose the **Offset** button from the Modify panel.
 - Or, at the 'Command:' prompt, enter the **offset** command:
 Command: **offset** (Press ENTER.)
 - Alternatively, enter the **o** alias at the 'Command:' prompt.
2. In all cases, AutoCAD displays the current settings:
 Current settings: Erase source=No Layer=Source OFFSETGAPTYPE=0
 And then prompts you for the offset distance:
 Specify offset distance or [Through/Erase/Layer] <Through>: (Enter a distance.)
3. Select the object to offset:
 Select object to offset or [Exit/Undo] <Exit>: (Select one object.)



4. Pick the side on which the offset should be placed:
 Specify point on side to offset or [Exit/Multiple/Undo] <Exit>: (Pick point 3.)
5. Press **ENTER** to exit the command:
 Specify point on side to offset or [Exit/Multiple/Undo] <Exit>: (Press ENTER.)

The “Select object to offset:” prompt repeats to allow you to offset as often as you wish, but just one object at a time. Press **ENTER** to terminate the command.

For the offset distance, you can pick two points on the screen, or enter a number representing the distance. If you get the message, “That object is not parallel with the UCS,” this means that the direction of the object’s Z axis is not parallel to the current user coordinate system.

MAKING OFFSET COPIES: ADDITIONAL METHODS

The **OFFSET** command’s **Through** option lets you specify a point through which the copy is offset. In addition, two system variables let you preset parameters.

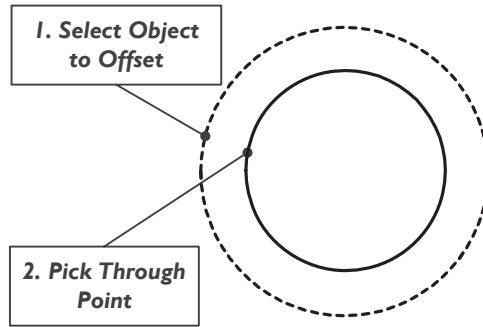
- **Through** option combines the distance and side options.
- **Erase** option erases the source object.
- **Layer** option specifies the destination layer.
- **OFFSETGAPTYPE** system variable determines how polyline gaps are handled.

Let’s look at each.

TUTORIAL: THROUGH

The **Through** option constructs the offset copy “through” a point. In effect, it combines the “Specify offset distance” and “Side to offset” options.

1. Draw a circle.
2. Start the **OFFSET** command, and then enter the **Through** option.
Command: **offset**
Specify offset distance or [Through]: **t**
3. Select the circle, and then pick the point through which the copy should be offset:
Select object to offset or <exit>: *(Pick the circle.)*
Specify through point: *(Pick point 1.)*



4. Continue making offset copies, or press **ENTER** to end the command.
Select object to offset or <exit>: *(Press ENTER.)*

Erase

The **Erase** option erases the source object:

Erase source object after offsetting? [Yes/No] <No>: *(Type Y or N.)*

“Yes” erases the source object, while “No” retains it. AutoCAD remembers this setting until you change it. (There is no system variable associated with it.)

Layer

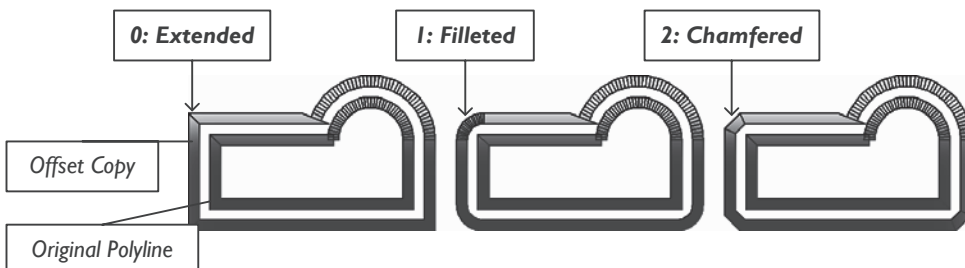
The **Layer** option specifies the destination layer:

Enter layer option for offset objects [Current/Source] <Source>: *(Type C or S.)*

“Current” places offset objects on the current layer, while “Source” places them on the same layer as the source object.

OffsetGapType

Offsetting polylines can be tricky. For this reason, AutoCAD includes the **OFFSETGAPTYPE** system variable to help you decide how potential gaps between polyline segments should be handled. The choices are extending lines, creating fillets (arcs), or creating bevels (chamfers).



Command: **offsetgaptype**

Enter new value for OFFSETGAPTYPE <0>: *(Enter a number between 0 and 2.)*

OffsetGapType	Comment
0	Gaps filled with extended line segments (default).
1	Gaps filled with filleted segments, creating arcs.
2	Gaps filled with chamfered line segments, creating beveled edges.

DIVIDE AND MEASURE

The **DIVIDE** and **MEASURE** commands place copies of points and blocks along objects.

DIVIDE divides an object into an equal number of parts, while **MEASURE** works with a specific distance — *number* of segments versus *length* of segment. Both commands place either blocks or points along the object.

(The **MEASURE** command has nothing to do with measuring distances or lengths; for those, use the **DIST** and **LIST** commands.)

Like the **OFFSET** command, dividing and measuring work with just one object at a time. Use the cursor to select a single object, because you cannot use Window, Crossing, or Last selection modes. In addition, AutoCAD is limited to working with lines, arcs, circles, splines, and polylines; picking a different object results in the complaint:

Cannot divide that object. * Invalid*

AutoCAD does not place a point or block at the start or end of open objects.



Note After using the **DIVIDE** and **MEASURE** commands to place points on an object, you can snap to the points with the **NODE** object snap.

To use the **DIVIDE** and **MEASURE** commands, you need to tell AutoCAD three things: (1) the object to be marked, (2) the number of markers, and (3) whether the markers are points or blocks.

TUTORIAL: DIVIDING OBJECTS

1. To see the effect of the **DIVIDE** command better, first change visibility of points.

Command: **pdmode**

Enter new value for PDMODE <0>: **4**

2. To divide an object into equal parts, start the **DIVIDE** command:

- At the 'Command:' prompt, enter the **divide** command:

Command: **divide** (Press ENTER.)

- Alternatively, enter the **div** alias at the 'Command:' prompt.

3. In all cases, AutoCAD prompts you to select the single object to divide:

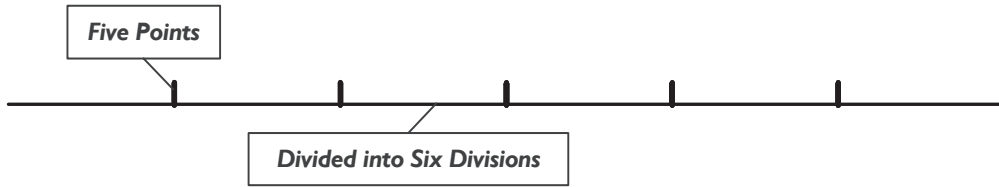
Select object to divide: *(Select one object.)*

4. Specify the number of divisions:

Enter the number of segments or [Block]: *(Enter a number between 2 and 32767.)*

The **DIVIDE** command results in evenly spaced points, and with one fewer point than you would expect: divide a line by *six*, and AutoCAD places *five* points, creating *six* divisions. (Count the number of

points and divisions on the line illustrated below.) The points and blocks are independent of the line; you can move and erase them at will.



The **MEASURE** command operates similarly to the **DIVIDE** command, the difference being that you specify the length of segment along which to space the points.

TUTORIAL: “MEASURING” OBJECTS

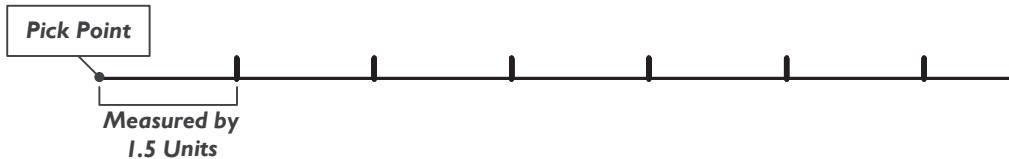
1. To place points along an object at specific distances, start the **MEASURE** command:
 - At the ‘Command:’ prompt, enter the **measure** command.

Command: **measure** (Press ENTER.)

- Alternatively, enter the **me** alias at the ‘Command:’ prompt
2. In all cases, AutoCAD prompts you to select the single object to divide:

Select object to measure: (Select one object.)
 3. Specify the number of divisions:

Specify the length of segment or [Block]: (Enter number higher than 1.)



This may sound arcane, but the **MEASURE** command needs a *starting point*, which depends on the object. As the figure above illustrates, measurement evenly distributes points as does division; there is usually a section left over at the end.

For open objects — lines, arcs, splines and open polylines — the starting point is the endpoint closest to your pick point; for closed polylines, it’s the point where you began drawing the polyline. For circles, it is at the current snap angle, which is usually 0 (at the circle’s 3 o’clock point); the measurement is made in the counterclockwise direction.

DIVIDING AND MEASURING: ADDITIONAL METHODS

Both commands’ **Block** option lets you specify a block to place along the object, in place of a point. And, since points tend to be invisible, the **PDMODE** and **PDSIZE** system variables (accessed by **DDPTYPE**) are useful.

- **Block** option places blocks along the object.
- **DDPTYPE** command changes the look of the points.

Let’s look at each.

Block

The **Block** option places blocks (symbols) along the object. The block must already exist in the drawing. The option operates identically for both commands:

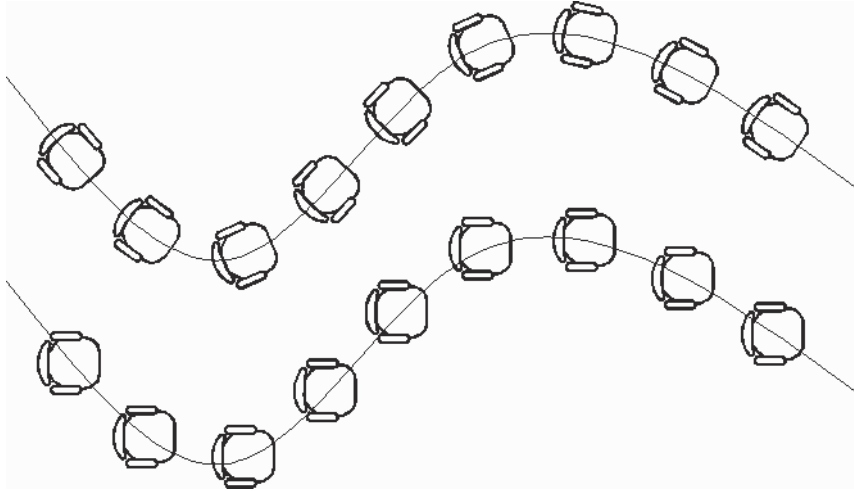
1. Start either command. Enter the **b** option, and then the name of a block:

Enter the number of segments or [Block]: **b**

Enter name of block to insert: *(Type name.)*

2. Decide whether you want the block aligned with the object (Y) or at its own orientation (N):
Align block with object? [Yes/No] <Y>: *(Enter Y or N.)*
3. Continue with the command.

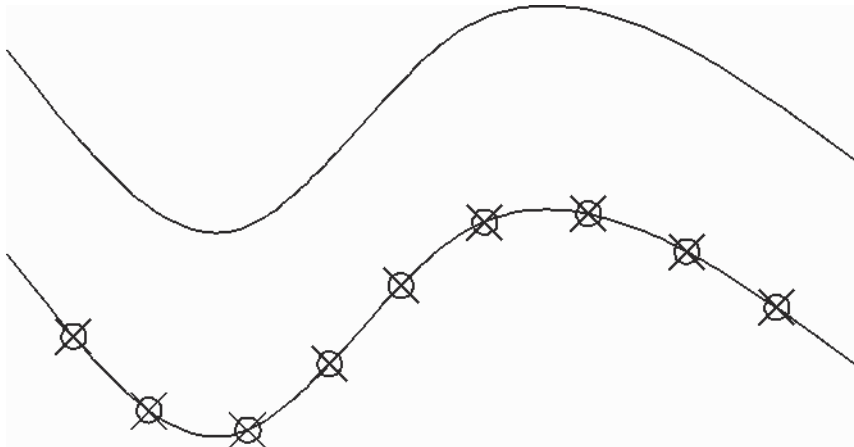
“Yes” means the inserted block turns with the divided object, such as arc, circle, or spline. “No” means the block is always oriented in the same direction. If you are unsure of the block’s name or even of its existence in the drawing, use the DesignCenter to help you.



Blocks aligned with a spline (top) and unaligned (bottom).

DdPType

Recall from Chapter 4, “Drawing with Basic Objects,” that the **DDPTYPE** command changes the look and size of points. Points are normally invisible (for all intents and purposes), so it may be useful to change their size. The figure below illustrates the before-and-after difference.



Spline divided by points of normal type (top), and enlarged (bottom).

ARRAY

The **ARRAY** command makes evenly-spaced copies in linear, rectangular, and round patterns.

There are times when you want to place multiple copies of object in patterns, like rows of seats in a movie theater, or columns of parking spaces at a shopping center. If you were using traditional drafting techniques, you would draw each one separately — tediously.

Already in the chapter, you saw how the **COPY** command (with its **Multiple** option) and the **DIVIDE** and **MEASURE** commands can place many copies in drawings. The **ARRAY** command, however, proves superior for making copies in precise rows, columns, matrices, circles, and semicircles. After you array an object, each copy can be edited separately — unlike the similar **MINSERT** command. For placing many copies in random places, however, **COPY** is better.

The largest number of rows and columns you can enter is 32,767; the smallest number is 1. Because a 32767 x 32767-array creates just over a billion elements, AutoCAD limits the total number to 100,000 — otherwise your computer system would overload. (The seemingly arbitrary value of 32,767 comes from one less than 2^{15}).

You can change the upper limit to another value between 100 and 10,000,000 with the **MAXARRAY** system registry variable. At the command prompt, enter "MaxArray" exactly as shown to set the limit to 1000:

Command: (**setenv "MaxArray" "1000"**)
"1000"

To create arrays, AutoCAD needs to know (1) the type of array, rectangular or polar, (2) the object(s) you plan to array, which must already exist in the drawing, and (3) the parameters of the array.

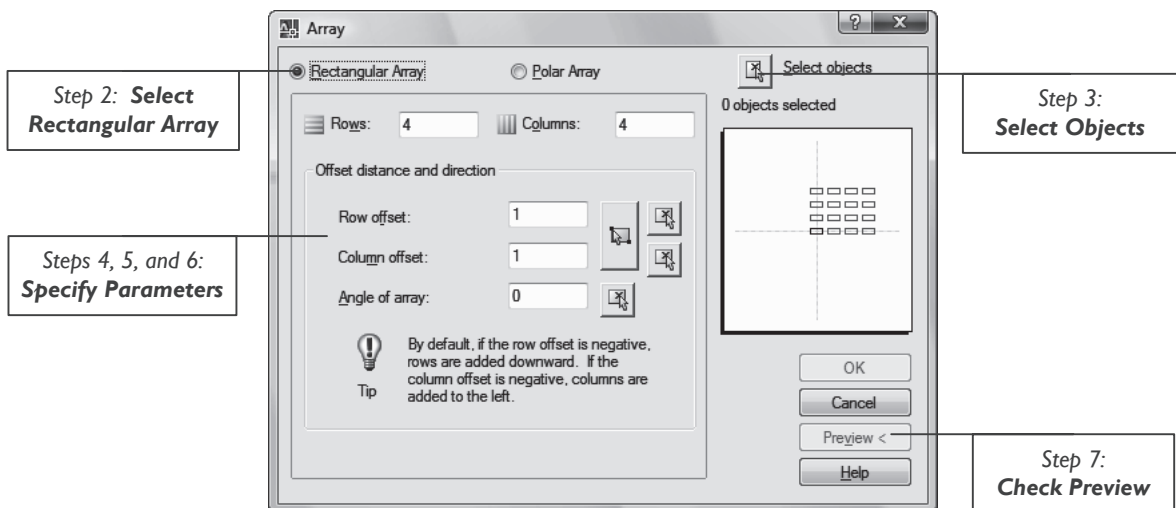
TUTORIAL: LINEAR AND RECTANGULAR ARRAYS

A linear array copies objects in the horizontal direction (in a row or the x direction) or the vertical direction (in a column or the y direction). A rectangular array copies the objects in a rectangular pattern made up of rows and columns.

- I. To make an array of copies, start the **ARRAY** command:
 - In the ribbons's 2D Home tab, choose the **Array** button from the Modify panel.
 - At the 'Command:' prompt, enter the **array** command:

Command: **array** (Press ENTER.)

- Alternatively, enter the **ar** alias at the 'Command:' prompt.



2. In all cases, AutoCAD displays the Array dialog box:
Notice that the dialog box has two radio buttons (the round buttons at the top of the dialog box) that determine the type of array:
 - **Rectangular** displays options for creating linear and rectangular arrays.
 - **Polar** displays options for creating polar (circular and semicircular) arrays.
3. Choose **Select Objects** to select the object(s) to array. The dialog box disappears, and this prompt appears:
Select objects: *(Select one or more objects.)*
Select objects: *(Press ENTER to return to the dialog box.)*



Notes You can avoid this step by selecting the objects before starting the **ARRAY** command.

Sometimes it's difficult to distinguish between *rows* and *columns*. Rows go side to side, while columns go up and down. Still puzzled? Look at the preview window, or click the **Preview** button for a sneak peak.

4. In the **Rows** and **Columns** text boxes, specify the number of copies to make in rows and columns. When you enter a 1 for both, AutoCAD complains,
Only one element; nothing to do.
because a 1x1 array consists of the original object only.
5. The **Row Offset** and **Column Offset** options measure the distance between the elements of the array. You can enter a specific distance, or click the adjacent buttons to select the distances in the drawing:
Specify the distance between rows (or columns): *(Specify a distance.)*
Enter a distance, or pick two points that specify the distance between row or column elements in the array. After picking the second point, the dialog box returns.

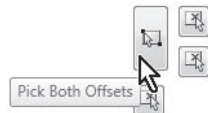
The **Pick Both Offsets** button clears the dialog box, and AutoCAD prompts you at the command line:

Specify unit cell: *(Pick a point.)*

Other corner: *(Pick another point.)*

Pick two points, creating a rectangle that specifies the row and column distance between elements in the array.

After you pick the second point, the dialog box returns.

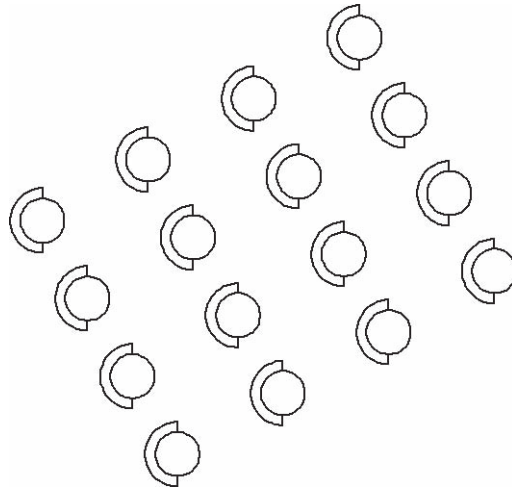


Notes AutoCAD normally creates rows to the right, and columns *upwards*. To have AutoCAD draw the array elements in the other direction, enter negative values for the row and column offsets.

If you use the mouse to pick two points, and the second point is below or to the left of the first, then AutoCAD automatically sets negative values.

To save file space in drawings with arrays, first turn the objects to be arrayed into a block, and then array the block.

6. **Angle of array** tilts the rectangular array at an angle; note that the object itself does not tilt, but the elements of the array are staggered by the angle. Entering an angle of 180 degrees draws the array downward. (To select the angle in the drawing, click the **Pick Angle of Array** button.)



A 4x4 rectangular array at an angle of 30 degrees.

7. Click **Preview** to see what the array will look like. (As of AutoCAD 2009, the dialog box is replaced by the command prompt.)
Pick or press Esc to return to dialog box or <right-click to accept array>: (Move the mouse, or enter an option.)

Drag the mouse to pan the drawing, useful for when the array extends beyond the border of the viewport.

Twirl the rollerwheel to zoom in and out.

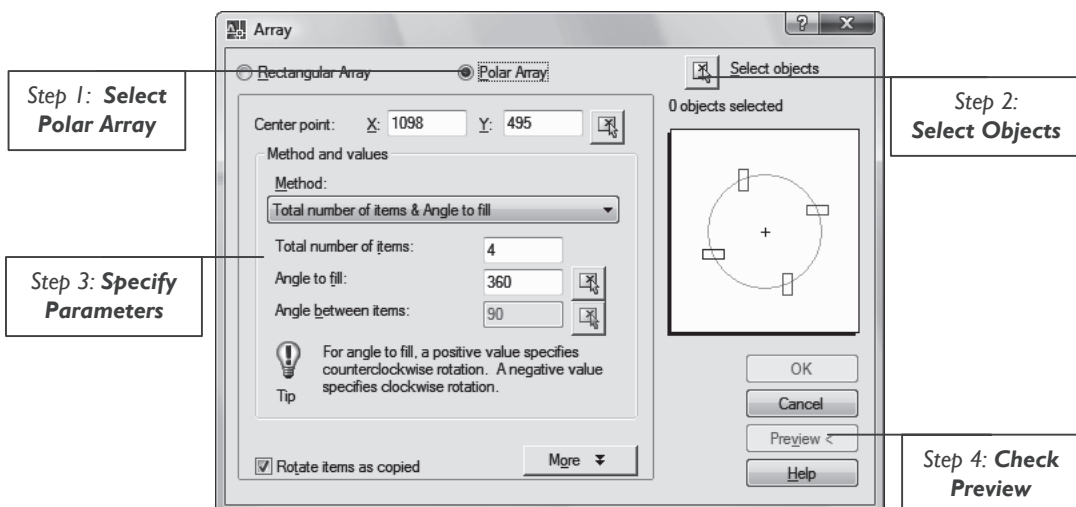
Click the drawing (or press **ESC**) to return to the dialog box.

Right-click to accept the array, and end the command.

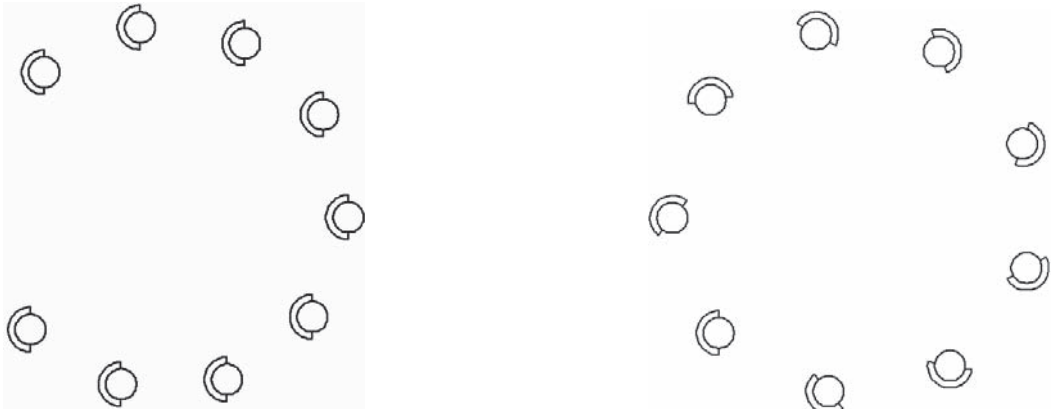
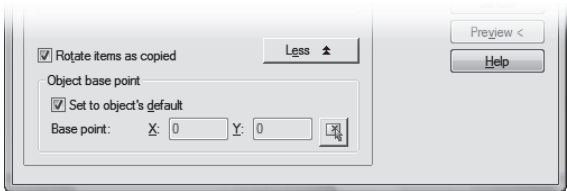
TUTORIAL: POLAR AND SEMICIRCULAR ARRAYS

Polar arrays arrange objects in circular patterns. To construct a polar array, you must define the angle between the items (from center to center, not actually between) and either the number of items or degrees to fill. You have the option of rotating (or not rotating) each object as it is arrayed.

1. Start the **ARRAY** command.
2. Click the **Polar Array** radio button. Now let's take a look at the options for creating a round array.



- 3. Select the **Center point** of the polar array; think of it as being the same as the center of a circle. AutoCAD automatically picks a point for itself using a method I have never figured out. (The technical editor agrees: "It's totally random!") But you can change it: enter new x, y-values or click the **Pick Center Point** button.
- 4. To create a polar array, you must provide data for any two of the following options. The Method droplist lets you pick the pair of options from the three possible choices:
 - **Total Number of Items** specifies the number of elements in the array. AutoCAD draws them to fit the polar route.
 - **Angle to Fill** changes the angle. By default, AutoCAD creates a 360-degree polar array, but you can create an arc array, instead. For example, specifying 270 degrees gives you three-quarters of a polar array.
 - **Angle Between Items** specifies the angle between the elements of the array.
- 5. Decide whether you want AutoCAD to **Rotate Items as Copied**. When turned on (the check mark appearing), AutoCAD makes sure the items "face the center."
- 6. Click the **More** button to see more options for polar arrays:



Left: 270-degree polar array with unrotated objects.
Right: 360-degree polar array with rotated objects.

The **Object base point** option is tricky to grok, but essentially you get to pick where AutoCAD begins to measure the distance from the object to the center of the polar circle. AutoCAD uses the following default values, which change depending on the object being arrayed:

Object	Default Base Point
Lines, polylines, donuts, 3D polylines, rays, splines	Starting point.
Ares, circles, ellipses	Center point.
Polygons, rectangles	First corner drawn.
Xlines	Midpoint.
Blocks, mtext, text	Insertion point.
Regions	Grip point.

- Click **Select objects** to pick one or more objects to array. Click **Preview** to see what the array will look like.

FILLET AND CHAMFER

The **FILLET** and **CHAMFER** commands create rounded and angled corners, respectively.

The **FILLET** command connects two lines or polylines with a perfect intersection, or with an arc of specified radius. Fillets can also connect two circles, two arcs, a line and a circle, a line and an arc, or a circle and an arc.

The two objects need not touch to be filleted — this includes parallel lines. This allows you to intersect two non-touching lines; in the case of parallel lines, a 180-degree arc is drawn between their ends.

To use the **CHAMFER** and **FILLET** commands, you need to tell AutoCAD two things: (1) the size of chamfer or fillet, and (2) the two objects to be chamfered or filleted.

Holding down the **SHIFT** key temporarily changes the fillet and chamfer distances to zero.

TUTORIAL: FILLETING OBJECTS

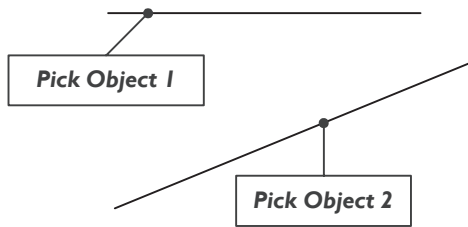
- To fillet a pair of objects, start the **FILLET** command:
 - In the ribbons's 2D Home tab, choose the **Fillet** button from the Modify panel.
 - At the 'Command:' prompt, enter the **fillet** command:

Command: **fillet** (Press ENTER.)
 - Alternatively, enter the **f** alias at the 'Command:' prompt.
- In all cases, AutoCAD first displays the current fillet settings, and then asks you to select two objects:

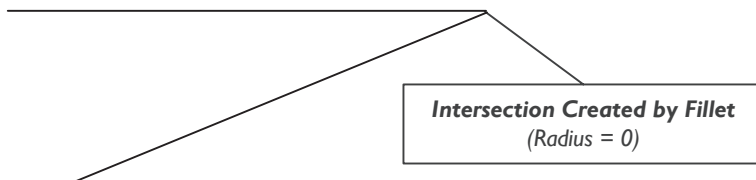
Current settings: Mode = TRIM, Radius = 0.0000

Select first object or [Undo/Polyline/Radius/Trim/Multiple]: (Pick object 1.)

Select second object or shift-select to apply corner: (Pick object 2.)



- And the command is done! Because the radius was set to 0 (the fresh-off-the-distribution-CD default value), AutoCAD creates a clean intersection, without the arc you might have been expecting. (Notice that AutoCAD extends the two lines so that they intersect.)

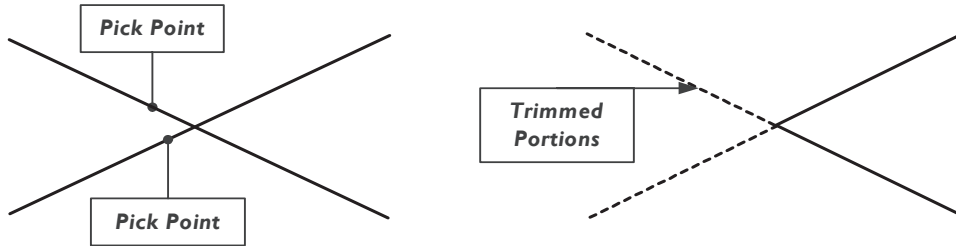


DIFFERENT FILLET RESULTS FOR DIFFERENT OBJECTS

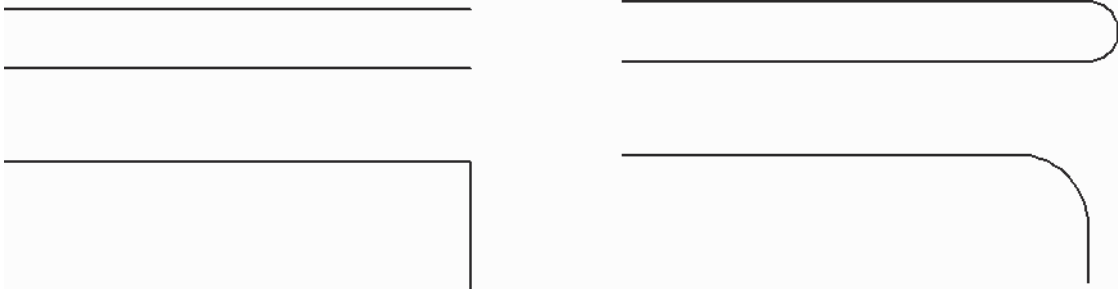
Depending on the objects involved, the fillet differs.

Lines

Two lines are trimmed back (or extended, as necessary), so that an arc fits between them. A zero-radius fillet connects two lines with a perfect intersection. The picked segments are trimmed off.



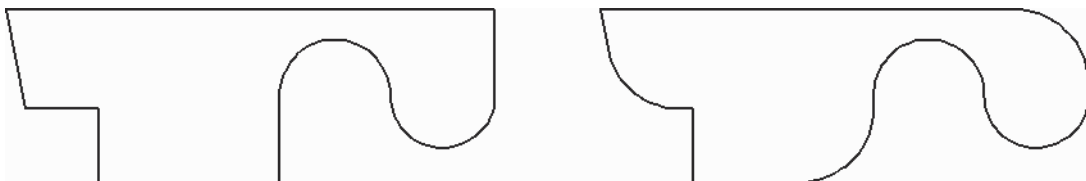
Two parallel lines fillet with a radius equal to their offset distance.



Two pairs of lines before and after being filleted with radius = 1.

Polylines

You can fillet an entire polyline in one operation when you select the **P** option (short for “polyline”). The fillet radius is placed at all vertices of the polyline. If arcs exist at any intersections, they are changed to the new fillet radius. Note that the fillet is applied to one continuous polyline.



Left: Original polyline.

Right: Filletted with the Polyline option;

notice that two vertices were too short to be filleted with radius = 1.

If the fillet radius is too large for a line or polyline segment, AutoCAD does not apply the fillet:

Select 2D polyline: (Select a single polyline.)

4 lines were filleted

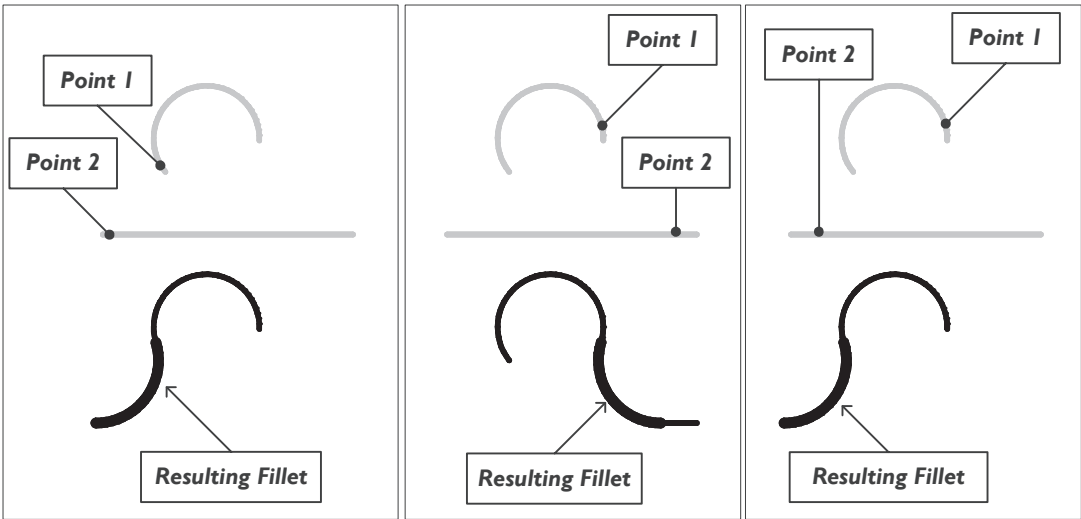
3 were too short

When a line and a polyline are filleted, all three objects (the line, the fillet arc, and the polyline) are converted to a single polyline.

Arcs and Circles

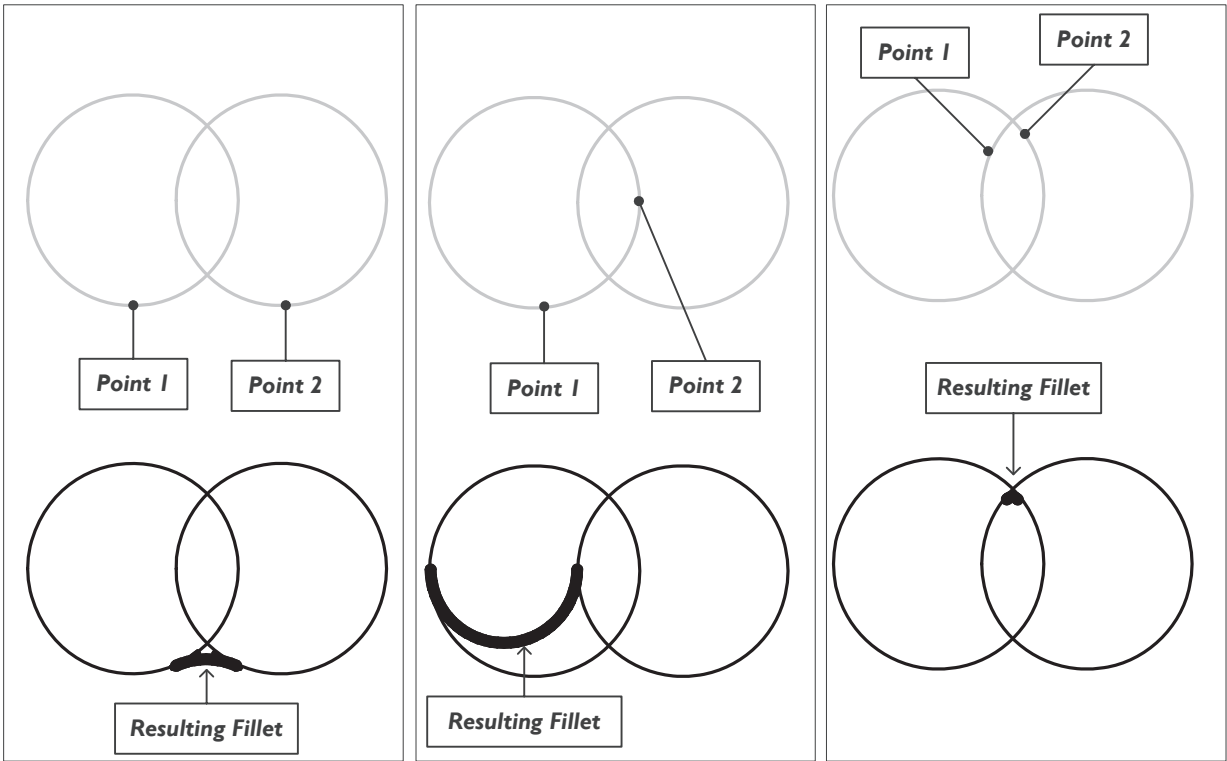
Lines, arcs, and circles can also be filleted. When you fillet such objects, however, there are often several possible fillet combinations. You specify the type of fillet by the points picked when you select the objects. AutoCAD attempts to fillet the endpoint closest to the selection point.

The figures illustrate several combinations between a line and arc. Observe the placement of the points used to pick the objects in the middle row, and the resulting fillet (shown in the bottom row).



The original line and arc shown in gray; the resulting fillet shown by the heavy arcs.

As with lines and arcs, the result of filleting two circles depends on the location of the two points you use to select the circles. The figure illustrates three possible combinations, each using different selection points.



The original pre-fillet circles shown in gray; the resulting fillet shown by the heavy arcs.



Notes When you select two objects for filleting, and get an undesirable result, use the **Undo** option to undo the fillet. Try to respecify points closer to the endpoints you want to fillet.

If you have several filleted corners to draw, construct your intersections at right angles, and then fillet each later. This allows you to continue the **LINE** command without interruption, and requires fewer commands.

Changing an arc radius by fillet is cleaner and easier than erasing the old arc and cutting in a new one. Let AutoCAD do the work for you!

You “clean up” line intersections by setting the fillet radius to zero and filleting the intersections.

CONSTRUCTING FILLETS: ADDITIONAL METHODS

The command has several options for special cases:

- **Undo** undoes the last fillet operation.
- **Polyline** treats polylines differently.
- **Radius** changes the radius of three-dimensional arrays.
- **Trim** determines what happens to the leftover bits.
- **Multiple** continues the command to fillet additional objects.

Let’s look at each.

Undo

The **Undo** option undoes the last fillet operation, without requiring you to exit the command to access the U or UNDO commands.

Polyline

The **Polyline** option fillets all vertices of a single polyline. When you fillet a polyline without using this option, AutoCAD expects you to place a fillet between two adjacent segments. At the “Select first object” prompt, select a single polyline.

Select 2D polyline: *(Select a single polyline.)*

Radius

The **Radius** option determines the radius of the fillet arc. When set to 0, the command ensures that the two lines match precisely; an arc is not created.

Specify fillet radius <1.0000>: *(Enter a radius.)*

If the fillet radius is too large for a line, AutoCAD complains:

Radius is too large *Invalid*

In that case, use a radius smaller than the shortest line.

Press **SHIFT** to override temporarily the radius to 0.0.

Trim

The **Trim** option determines what happens to the trimmed bits. When on (the default), the command trims away the selected edges, up to the fillet arc’s endpoint. When off, no trim occurs.

Enter Trim mode option [Trim/No trim] <Trim>: *(Enter T or N.)*

Multiple

The **Multiple** option (formerly **mUltiple**) repeats the command until you press **ESC**:

Select first object or [Undo/Polyline/Radius/Trim/Multiple]: *(Press esc to exit command.)*

CHAMFER

The CHAMFER command trims segments from the ends of two lines or polylines, and then draws a straight line or polyline segment between them. The distance to be trimmed from each segment can be different or the same. The two objects do not have to intersect, but they must be capable of intersecting if there were extended. Unlike fillets, parallel lines cannot be chamfered.

Chamfering only works with line segments, such as lines, 2D polylines, and traces. It does not work with arc segments, such as arcs, circles, and ellipses.

TUTORIAL: CHAMFERING OBJECTS

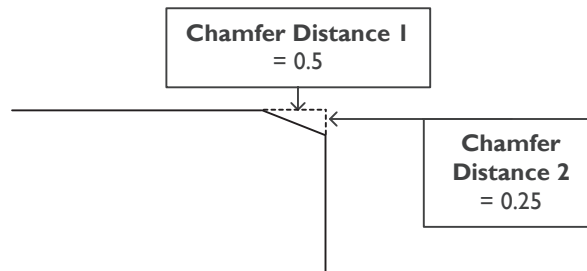
- To chamfer a pair of objects, start the **CHAMFER** command:
 - In the ribbon's 2D Home tab, choose the **Chamfer** button from the Modify panel.
 - At the 'Command:' prompt, enter the **chamfer** command:

Command: **chamfer** (Press ENTER.)
 - Alternatively, enter the **cha** alias at the 'Command:' prompt.
- In all cases, AutoCAD first displays the current chamfer settings:

(TRIM mode) Current chamfer Dist1 = 0.0000, Dist2 = 0.0000
- Before selecting objects, change the chamfer distance from the current setting of 0:

Select first line or [Undo/Polyline/Distance/Angle/Trim/mEthod/Multiple]: **d**

Specify first chamfer distance <0.0000>: (Enter a value, such as **.5**.)

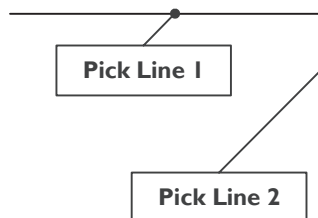


- Chamfers work with two distances, which can be different. Enter the second distance:

Specify second chamfer distance <0.5000>: (Enter another value, such as **.25**.)
- Now pick the two lines to chamfer:

Select first line or [Undo/Polyline/Distance/Angle/Trim/mEthod/Multiple]: (Pick line 1.)

Select second line or shift-select to apply corner: (Pick line 2.)



AutoCAD creates a chamfer at the intersection.

You can specify the amount to be trimmed by entering either a numerical value, or by showing AutoCAD the distance using two points on the screen.

CONSTRUCTING CHAMFERS: ADDITIONAL METHODS

The command has several other options for special cases:

- **Undo** undoes the last chamfer operation.
- **Polyline** chamfers a single polyline.
- **Angle** specifies the chamfer angle.
- **Trim** determines whether end pieces are saved.
- **mEthod** specifies the chamfer method.
- **Multiple** continues the command to fillet additional objects.

Let's look at each.

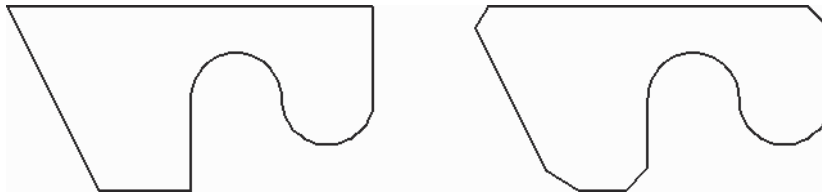
Undo

The **Undo** option undoes the last chamfer operation, without requiring you to exit the command to access the U or UNDO commands.

Polyline

Like filleting, chamfering a polyline is different from chamfering a pair of lines. If you had used the **PLINE** command's **Close** option to finish the polyline, AutoCAD chamfers *all* corners of the polyline; if not, the final vertex is not chamfered.

Select 2D polyline: (*Pick a polyline.*)



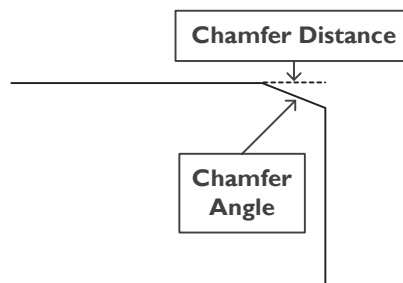
Left: Polyline before being chamfered.

Right: Chamfered with the **Polyline** option; notice that arc vertices are not chamfered.

If the polyline contains arcs, they are not chamfered. If some parts of the polyline do not chamfer, it could be that the segments are too short or parallel to each other. In that case, AutoCAD warns: “n were too short.”

Angle

As an alternative to specifying a chamfer by two distances, you can specify a distance and an angle.



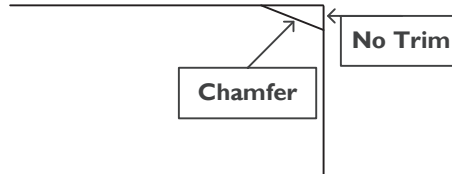
Specify chamfer length on the first line <1.0000>: (*Enter the distance.*)

Specify chamfer angle from the first line <0>: (*Enter the angle.*)

Trim

Normally, the **CHAMFER** command erases the line segments not needed after the chamfer. The **Trim** option, however, determines whether you keep the excess lines.

Enter Trim mode option [Trim/No trim] <Trim>: (Enter **T** or **N**.)



mEthod

The default method specifies two distances. If you prefer the distance-angle method, the **mEthod** option (formerly **Method**) lets you change the default:

Enter trim method [Distance/Angle] <Angle>: (Type **D** or **A**.)

Multiple

The **Multiple** option (formerly **mUltiple**) repeats the command until you press ESC:

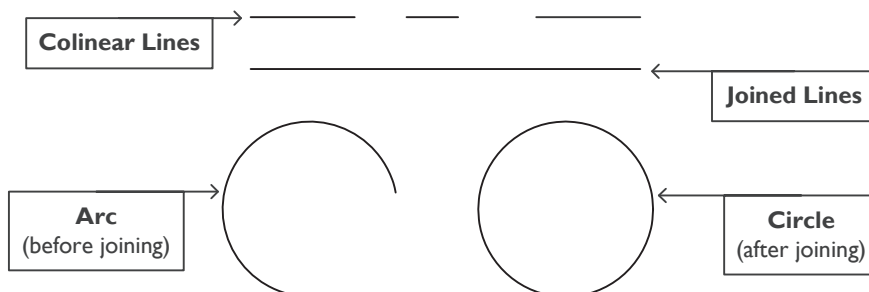
Select first object or [Polyline/Radius/Trim/mUltiple]: (Press **esc** to exit command.)

JOIN

The **JOIN** command joins similar objects into one.

The following objects can be joined, subject to these conditions:

- **Colinear lines** joined into a single line, including those with overlaps and gaps between them. (*Colinear* means the lines are lined up in a row like cars in a train.)
- **Arcs** in the same imaginary circle joined into a single arc counterclockwise from the source arc; they can have overlaps and gaps between them. This command also closes arcs into circles. When arcs are dimensioned with **DIMARC**, the converted circles are disassociated from the dimensions.
- **Elliptical arcs** in the same imaginary ellipse joined into a single elliptical arc counterclockwise from the source arc; they can have gaps between them. This command also closes elliptical arcs into ellipses.
- **Polylines**, lines, and arcs joined into a single polyline; they *cannot* have gaps between them. When the first object selected is a polyline, then this works like the **PEDIT Join** command.
- **Splines** joined into a single spline; they *cannot* have gaps between them; splines and helixes can be joined to each other (new to AutoCAD 2010).
- The source object determines the properties of the joined objects.



To use the **JOIN** command, you need to tell AutoCAD two things: (1) the source object, and (2) the other objects to join; when arcs and elliptical arcs are selected, there is also the choice of whether you want the open objects closed.

TUTORIAL: JOINING OBJECTS

- To join two or more objects, start the **JOIN** command:
 - In the ribbons's 2D Home tab, choose the **Join** button from the Modify panel.
 - At the 'Command:' prompt, enter the **join** command:

Command: **join** (Press ENTER.)
 - Alternatively, enter the **j** alias.
- AutoCAD prompts you to select the source object, which can be a line, arc, elliptical arc, polyline, or spline:

Select source object: (Select one object.)
- AutoCAD prompts you to select the objects to join. The prompt varies, depending on the source object selected; the following prompt is for arcs:

Select arcs to join to source or [cClose]: (Select one or more objects.)
- AutoCAD repeats the prompt until you press **ENTER** to exit the command:

Select arcs to join to source or [cClose]: (Press ENTER.)

JOINING OBJECTS: ADDITIONAL METHODS

The **JOIN** command has one option: **cClose** closes arcs and elliptical arcs.

Command: **join** (Press ENTER.)
 Select source object: (Select one arc or ellipse.)
 Select arcs to join to source or [cClose]: **c**
 Arc converted to a circle.



Note Use the **BREAK** command to convert circles into arcs; use the **JOIN** command to convert arcs into circles.

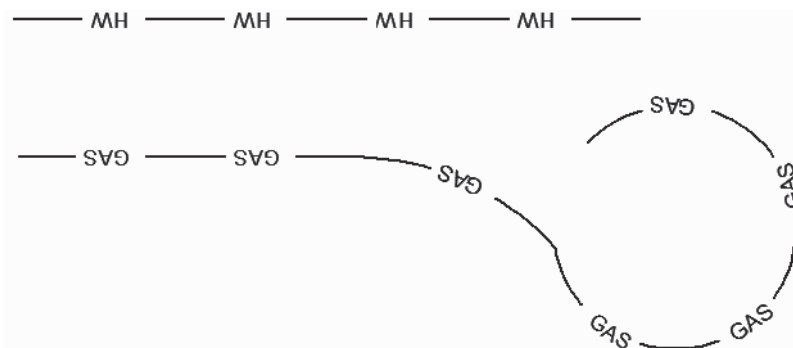


REVERSE

The **REVERSE** command reverses the direction of lines, polylines, splines, and helixes.

TUTORIAL: REVERSING OBJECTS

- From the Companion CD, open the Reverse.dwg file, a drawing containing lines with linetypes.



2. To see the effect of reversing directions of objects, start the **REVERSE** command:
 - In the ribbon's 2D Home tab, choose the **Reverse** button from the Modify panel.
 - At the 'Command:' prompt, enter the **reverse** command:

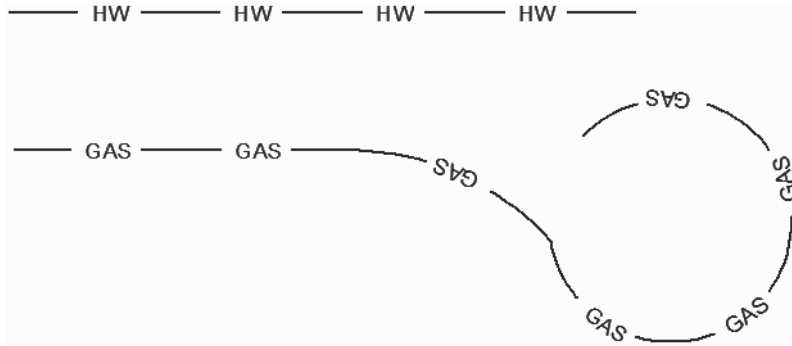
Command: **reverse** (Press ENTER.)

3. AutoCAD prompts you to select one or more objects, which can be lines, polylines, splines, and/or helices:

Select objects: (Select the lines.)

4. AutoCAD repeats the prompt until you press **ENTER** to exit the command:

Select objects: (Press ENTER.)



Notice that the linetype text has switched orientation, indicating the lines are reversed.



REVLOUD

The **REVLOUD** command creates revision clouds.

Revision clouds are often used to highlight areas in drawings that require attention, such as a revision or a potential error. Revision clouds are sometimes called “markups” or “redlines,” because they were often drawn with red pencils to stand out in drawings.

AutoCAD's **REVLOUD** command creates revision clouds, or converts other objects into revision clouds — specifically circles, ellipses, closed polylines, and closed splines. Because you cannot invoke transparent zooms and pans (other than through the mouse wheel) during the command, ensure you can see the entire area before starting.



Note Before starting **REVLOUD**, switch to a layer set to red. That makes it easier to turn on and off the display of the revision cloud, and makes it the traditional color of red to boot.

To use the **REVLOUD** command, you need to tell AutoCAD two things: (1) the starting point, and (2) the cloud path.

TUTORIAL: REDLINING OBJECTS

1. To markup drawings, start the **REVLOUD** command:
 - In the ribbon's 2D Home tab, choose the **Revision Cloud** button from the Draw panel.
 - At the 'Command:' prompt, enter the **revcloud** command.

Command: **revcloud** (Press ENTER.)

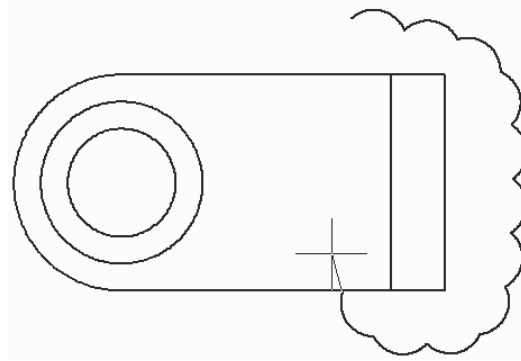
2. AutoCAD displays the current settings, and then prompts you to start drawing the revision cloud:

Minimum arc length: 0.5000 Maximum arc length: 0.5000 Style: Normal

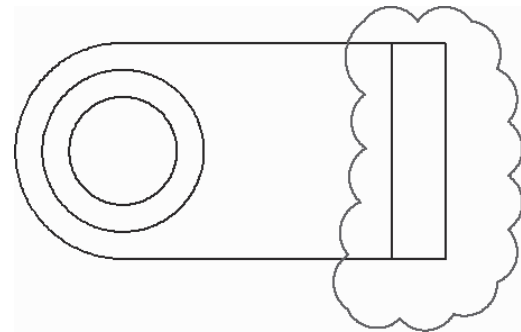
Specify start point or [Arc length/Object/Style] <Object>: (Pick a point.)

3. Move your cursor. Notice that AutoCAD automatically creates the cloud pattern. As an alternative, you can define the size of arcs by picking points.

Guide crosshairs along cloud path...



4. When the cursor is close to the start point, AutoCAD automatically closes the cloud: Revision cloud finished.



After placing the revision cloud, you can edit it like other objects in the drawing — through copying, grips editing, erasing, and so on.

REDLINING OBJECTS: ADDITIONAL METHODS

To help you in redlining, AutoCAD has these options:

- **Arc length** defines the size of the arcs making up the clouds.
- **Object** converts existing objects into revision clouds.
- **Style** switches between simple and calligraphic arcs.

Let's look at each.

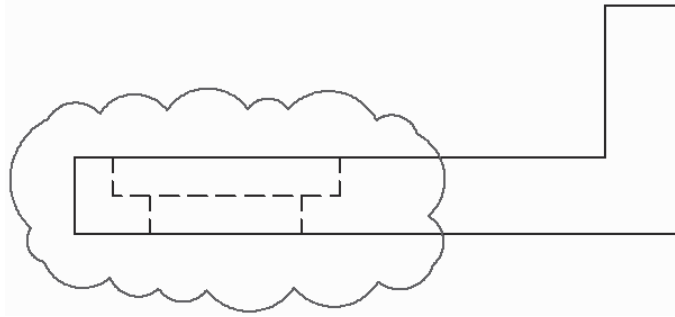
Arc length

The **Arc length** option determines the size of the arcs making up the clouds, which are scale-dependent, like linetypes and hatch patterns. **REVCLOUD** saves the arc length as a factor of the **DIMSCALE** system variable, so that clouds drawn after the dimension scale changes still look the right size when the scale factor changes.

When you specify different minimum and maximum lengths, AutoCAD draws clouds with random-size arcs. AutoCAD limits the maximum arc length to three times the minimum length.

Specify minimum length of arc <0.5000>: (Enter a value.)

Specify maximum length of arc <0.5000>: (Enter a value.)



A revision cloud with arcs of varying sizes.

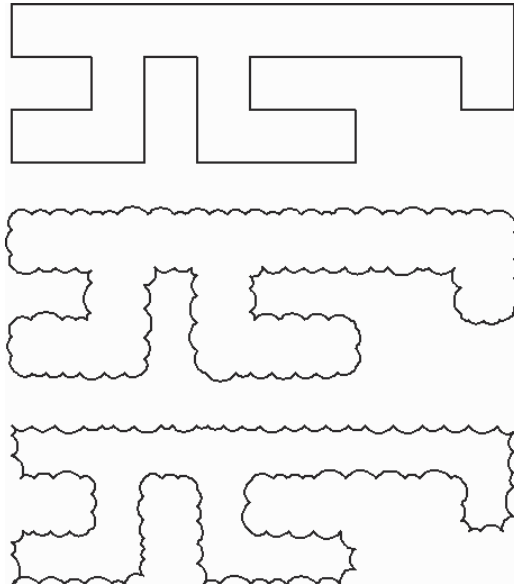
Object

The **Object** option lets you convert existing objects into revision clouds.

Select object: (Select one object.)

Reverse direction [Yes/No] <No>: (Enter **Y** or **N**.)

Revision cloud finished.



Top: The original polyline object.

Middle: Object converted to revision cloud.

Bottom: Revision cloud with reversed arcs.

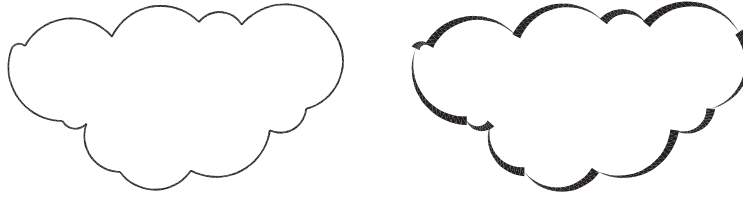
You can convert closed polylines, circles, ellipses, and closed splines to revision clouds. After applying this option, what happens to the original object? That depends on the setting of the **DELOBJ** system variable. If set to 1 (the default), the original is erased; if set to 0, the original object stays in the drawing.

Command: **delobj**

Enter new value for DELOBJ <1>: (Enter **1** or **0**.)

Style

The **Style** option switches between simple and calligraphic arcs. Calligraphic arcs, added in AutoCAD 2005, have a variable width, as illustrated below.



Left: Normal arcs.
Right: Calligraphic arcs.

To switch between normal and calligraphic arcs, use the **Style** option, as follows:

Specify start point or [Arc length/Object/Style] <Object>: **s**

Select arc style [Normal/Calligraphy] <Normal>: **c**

Arc style = Calligraphy

To convert calligraphic arcs to normal arcs, use the **PEDIT** command's **Width** option.



MARKUP

The **MARKUP** command imports marked-up *.dwf* files from Design Review (formerly DWF Composer; DWF is short for “design Web format”). It displays the Markup Set Manager, which places and edits redline DWF markups in drawings.

Despite its name, **MARKUP** does not mark up drawings; its purpose is to manage marked up drawings; the markup process takes place in Design Review. This command works only with *.dwf* files containing markup data; it will not open any other kind of markup file, nor does it “import” non-marked-up DWF data, or work with drawings marked up with the **REDLINE** or **RMLIN** commands.

(History: The **MARKUP** command replaces the **RMLIN** command [short for “Red Markup Line In”], which imported XML-format *.rml* markup files created by Autodesk’s old Volo View drawing viewer software. **RMLIN** was removed from AutoCAD 2006.)

Design Review

To use the **MARKUP** command, you must have access to Design Review, the free viewing, redlining, and printing software from Autodesk. You can download a copy from www.autodesk.com/designreview, following registration.



Design Review acts as a publisher, merging files from many sources — such as raster images, word processing documents, spreadsheets, and drawings from other CAD packages — into a single, multi-page document. To convert non-AutoCAD documents into DWF format, download and install the free DWF Writer printer driver. More information is available at www.autodesk.com/dwf.

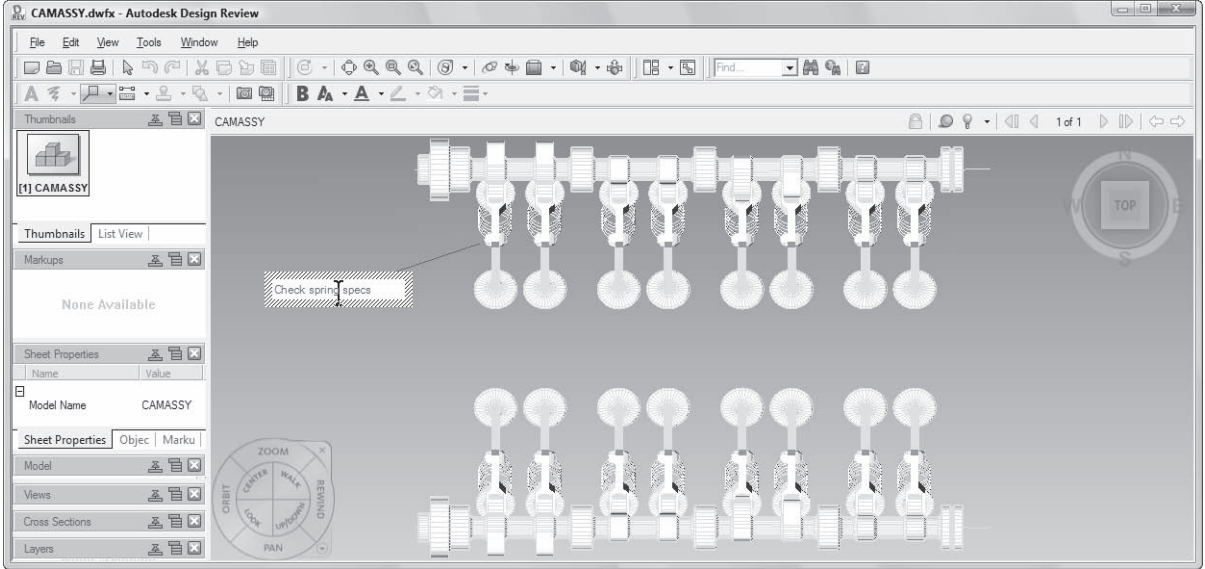
The markup cycle works like this:

In AutoCAD:

1. Create and edit the drawings.
2. Export the drawing from AutoCAD in DWF or DWFx format using the **PLOT**, **PUBLISH**, **AUTOPUBLISH**, **EXPORTDWF**, **EXPORTDWFX**, or **3DDWF** command. (See Chapter 20 for details.)

In Design Review:

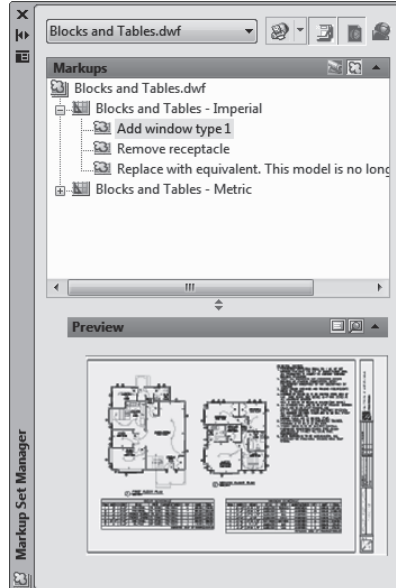
3. Open the .dwf file in Design Review.
4. Mark up the drawings using the markup tools found on the toolbar.



5. Save the markups with the **File | Save** command.

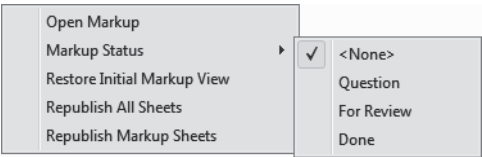
Back in AutoCAD:

6. Open the marked-up .dwf files with the **MARKUP** command.
7. To view the markups, double-click the markup name in the Markup Set Manager window, "Markup1" in the figure below.



Markup Options

Many of the MARKUP command’s options are “hidden” in shortcut menus. Depending on where you right-click in the Markup Set Manager palette, you get menus with differing sets of commands. The most-commonly used shortcut menu is found by right-clicking a markup sheet (indicated by the red revision cloud icon).



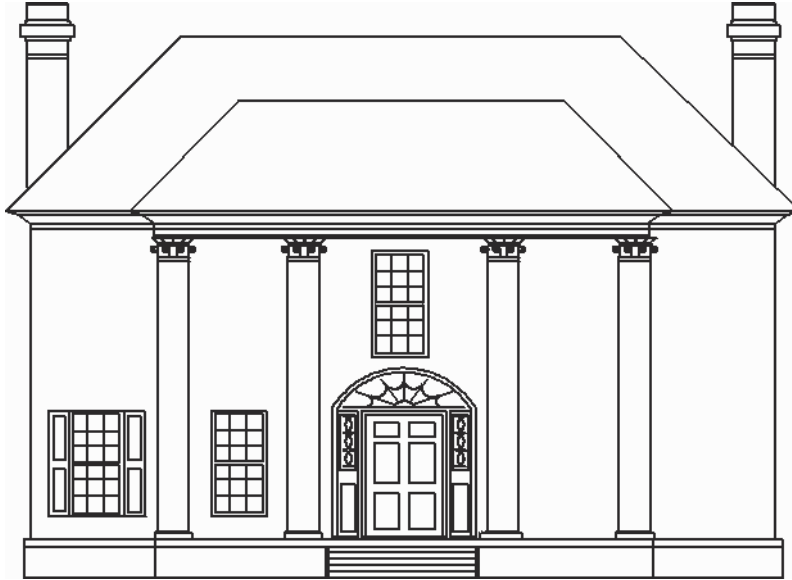
The menu’s options are:

Command	Comment
Open Markup	Finds and opens the original <i>.dwg</i> drawing file, and makes active the layout associated with the markup; the <i>.dwf</i> file is also opened, but is hidden until you press ALT+4.
Markup Status	Allows you to change the status of markups: <None> has no status. Question indicates additional information is needed. For Review indicates changes should be reviewed by another. Done indicates the markup is implemented.
Restore Initial Markup View	Restores the original markup view.
Republish All Sheets	Reoutputs all sheets in DWF format with the changes made.
Republish Markup Sheets	Overwrites the previous <i>.dwf</i> file with changes made to the drawing and the markup status.

Marked-up *.dwf* files can also be opened in AutoCAD with the OPENDWFMARKUP command. The MARKUPCLOSE command closes the Markup Set Manager window.

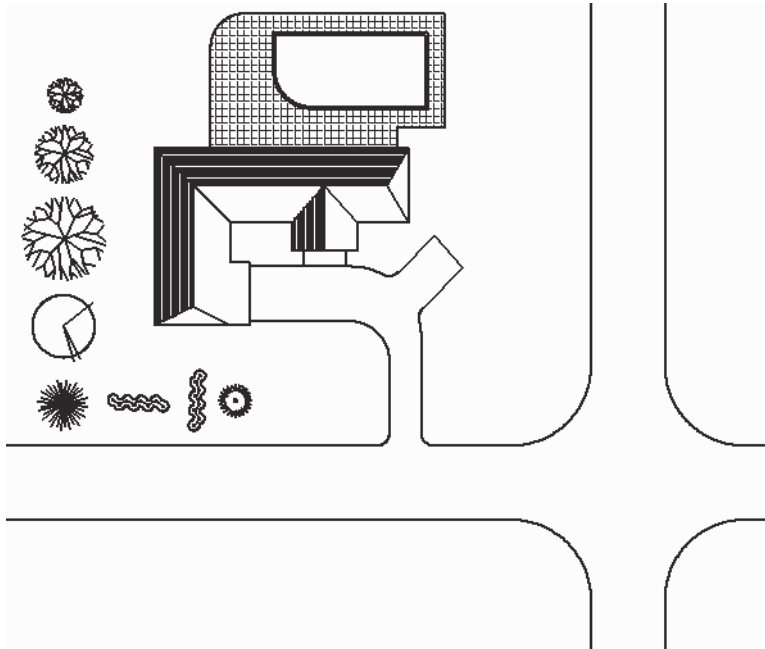
EXERCISES

1. From the Companion CD, open the *edit3.dwg* drawing file, an architectural elevation.



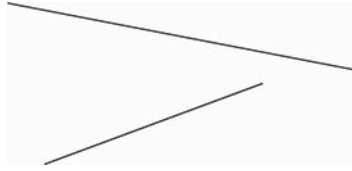
Use the **COPY** command to copy the windows from the left side to the right side. Then, copy all the windows on the lower level (including those you just copied) to the upper level. When copying, use the object selection options you think will work best.

2. From the Companion CD, open the *edit4.dwg* drawing file, a landscaping plan.

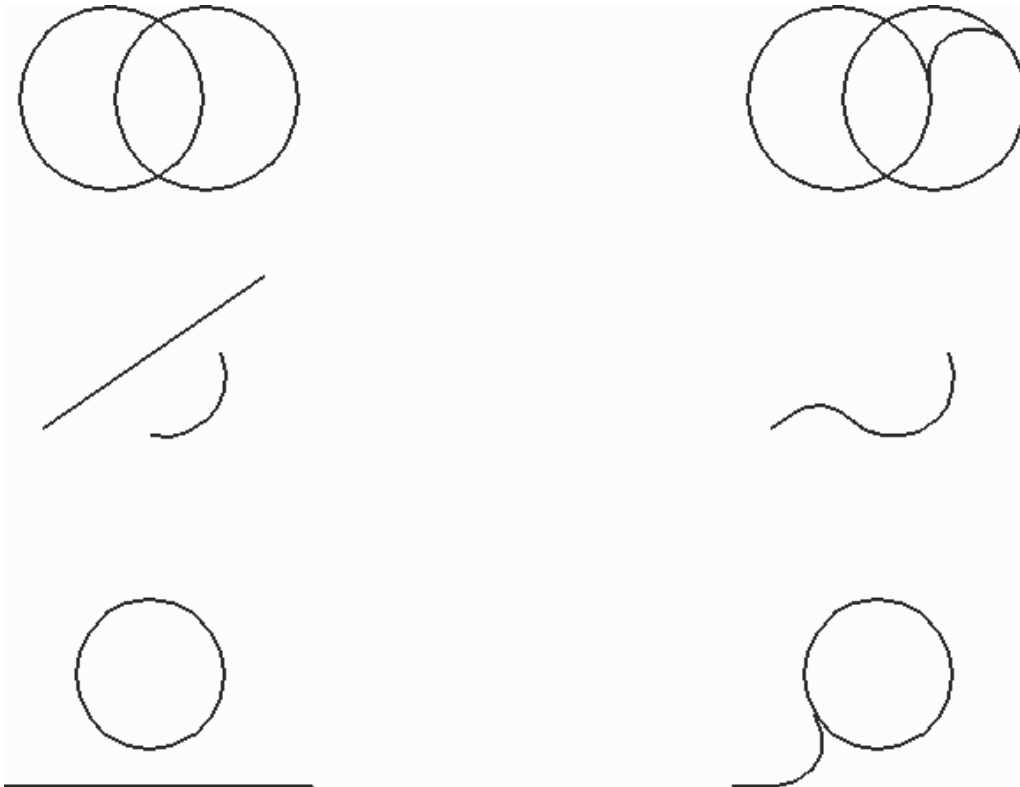


Use the **COPY** command to copy the landscaping blocks (trees, shrubbery, and so on) and to create a landscape scheme.

3. Connect two lines with fillets of varying radius.
First, draw lines similar to those in the illustration.



- a. Connect the two lines with a fillet of radius of 0.15.
 - b. Use the **SHIFT** key to set the fillet radius to zero, and then apply the fillet to the two lines again. Do the two lines now connect in a perfect intersection?
4. From the Companion CD, open the *edit6.dwg* drawing file, a practice drawing.
Fillet the objects to achieve the results shown on right side of the figure.



5. Suppose that you have to add another circle to your drawing that is identical to an existing circle.
Draw a circle, and then use the **COPY** command to place a second circle.
Is the second circle an exact copy of the first?

6. In this exercise, you create a rectangular array.

Start a new drawing.

As a visual aid, turn on the grid with a value of one.

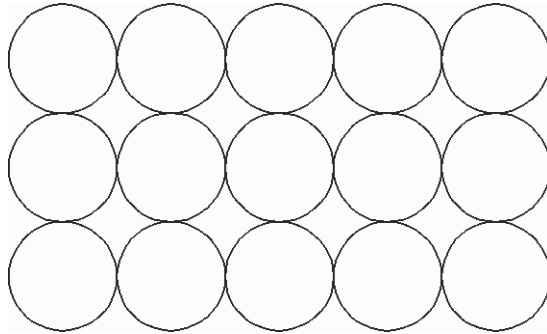
Draw a circle with the center point located at 1,1 and a radius of one.

Select the circle, and then start the **ARRAY** command.

In the dialog box, enter the following options:

Type of array	Rectangular
Rows	3
Columns	5
Row offset	2
Column offset	2

Your array should look like the following illustration:



Save the drawing with the name *array.dwg*.

7. Start another drawing to create a polar array.

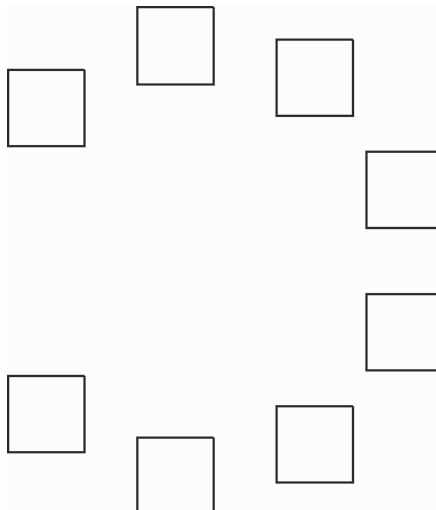
Draw a square with sides of one unit each.

Start the **ARRAY** command, and enter these options:

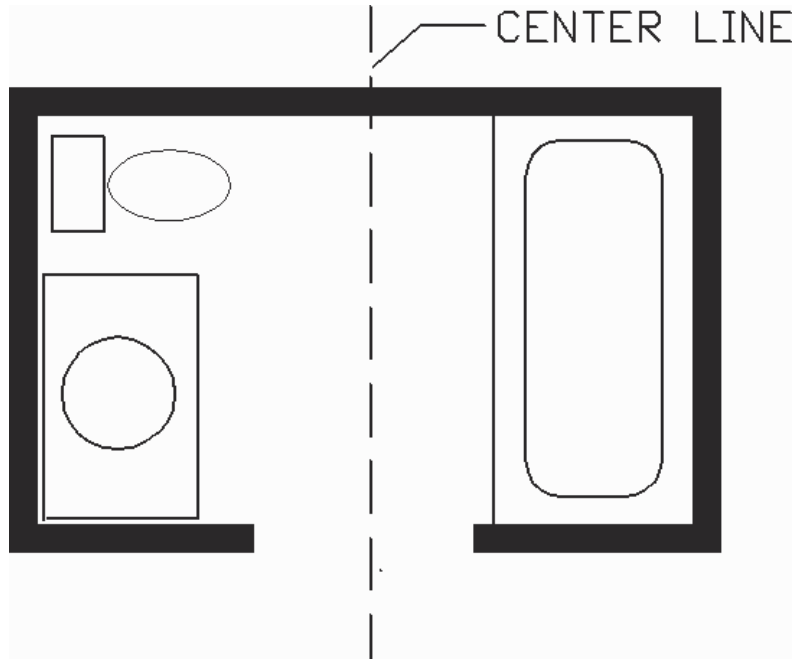
Select objects	<i>Select the square</i>
Type of array	Polar
Center point	2,2
Total number of items	4
Angle to fill	270
Rotate items as copied	off

The polar array should look like the one in the figure.

Save the drawing with the name *parray.dwg*.



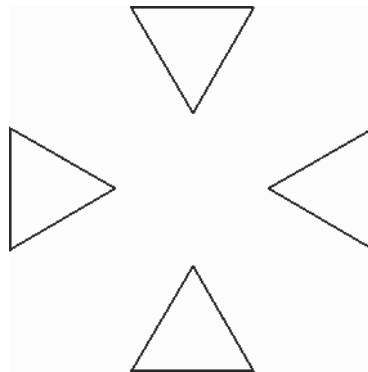
8. From the Companion CD, open the *mirror1.dwg* drawing file, a floor plan. Suppose that you are designing a house, and you want to reverse the layout of the bathroom.



Reverse the room using the **MIRROR** command.

Hint: Turn on ortho mode (**F8**), and delete the source objects.

9. In this exercise, the **PDMODE** system variable allows you to see points placed by the **DIVIDE** command.
 Start a new drawing.
 Draw a circle: center the circle in the viewport, and use a radius of 3.
 Start the **DIVIDE** command, and use 8 for the number of segments.
 Do you see any difference to the circle?
 Set the **PDMODE** system variable to 34.
 If necessary, use the **REGEN** command to make the new point style visible.
10. Draw a symbol similar to the one in the figure.
 Which two commands make it easy to draw the symbol?



Turn the symbol into a block, and name it "Symbol."

- a. Draw a circle on the screen, and then start the **DIVIDE** command.

Use 8 segments, along with the **Block** option, with blocks aligned with the object.

- b. Repeat the exercise, but this time don't rotate the block.

11. Let's "measure" an object.

Draw a horizontal line 6 units in length.

Set the point mode to 34 to make the points visible.

With the **MEASURE** command, specify a segment length of 1.0.

12. In this exercise, you use the **ELLIPSE** command to draw a can.

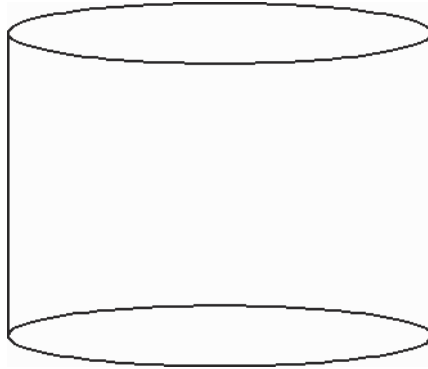
Draw one ellipse with the **ELLIPSE** command. Don't worry about the size.

Next, use the **COPY** command to copy the ellipse to create the top of the can.

Hint: Use ortho mode to align the copy perfectly with the original ellipse.

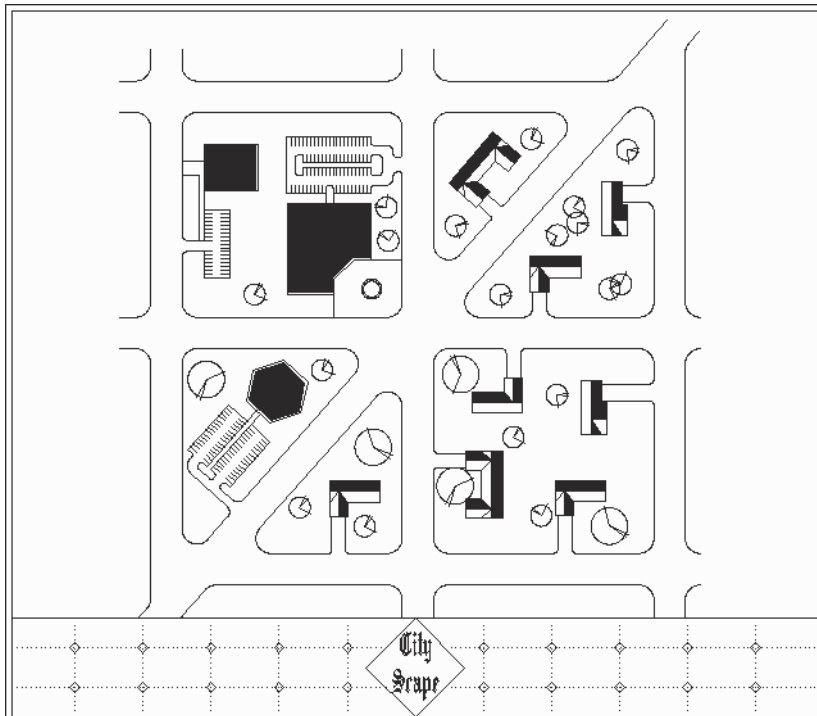
Now use the **LINE** command to draw lines between the two ellipses, as shown in the figure.

Hint: Use QUADrant object snap to capture the outer quadrants of the ellipses.



13. In this exercise, you use the **OFFSET** command to help draw a cityscape plan.

From the Companion CD, open the *offset.dwg* drawing file.

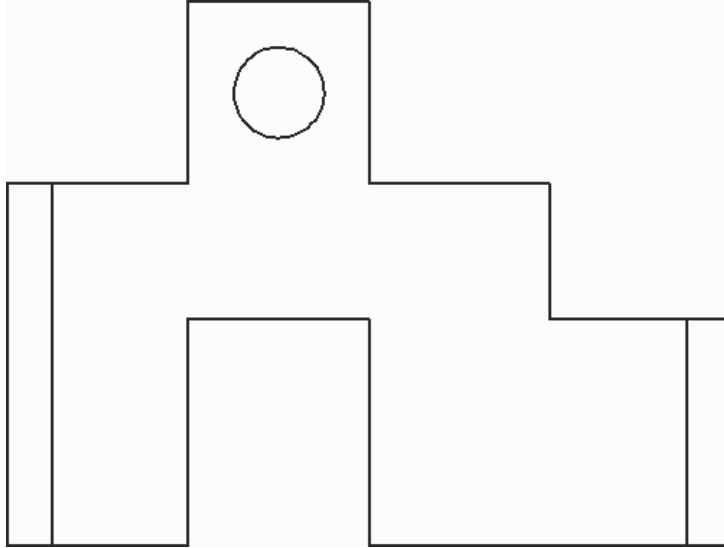


The edges of the streets are drawn with polylines.

Use the **OFFSET** command's **Through** option to offset the curbs by 6".

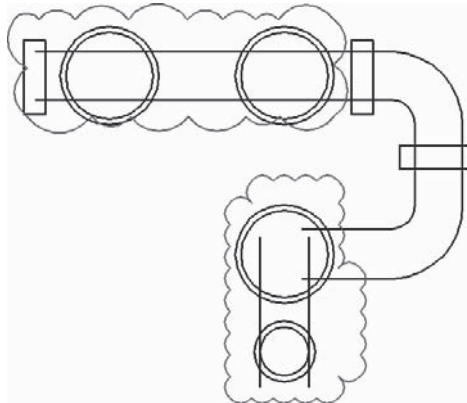
14. Draw the object shown in the figure below.

Use the **CHAMFER** command to bevel all its corners.



15. From the Companion CD, open the *edit5.dwg* drawing, a piping diagram.

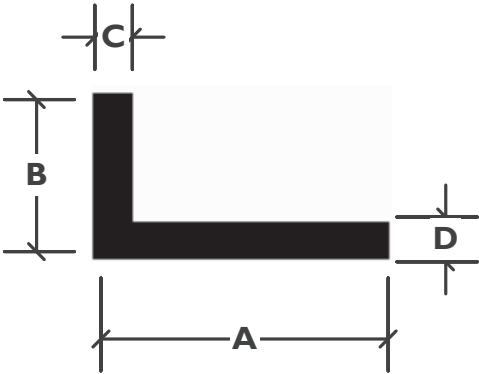
Use the **REVLOUD** command to place revision clouds around the two unfinished areas, as illustrated by the figure.



16. Draw the profiles of the aluminum extrusions diagrammed and dimensioned below. Apply fillets, where required, and hatch with a solid fill. All dimensions are in inches.

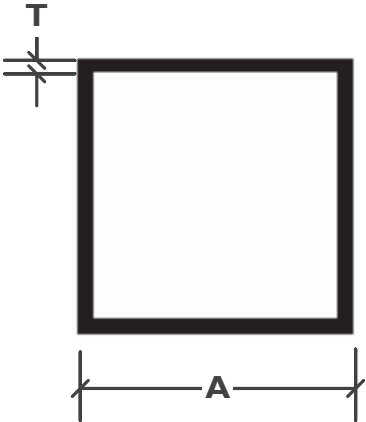
a. Angle.

A	B	C	D
4.000	1.000	0.155	0.125



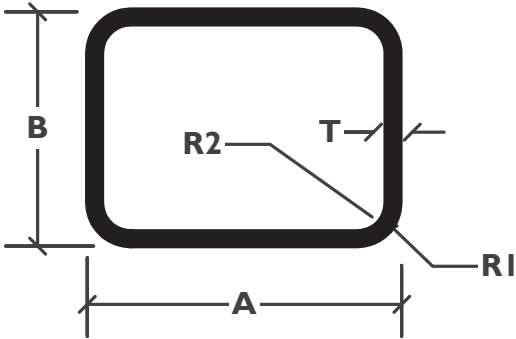
b. Square tube.

A	T
4.000	0.145



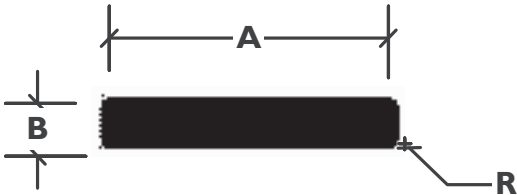
c. Rectangular tube with rounded corners.

A	B	T	R1	R2
2.250	1.750	0.125	0.125	0.125



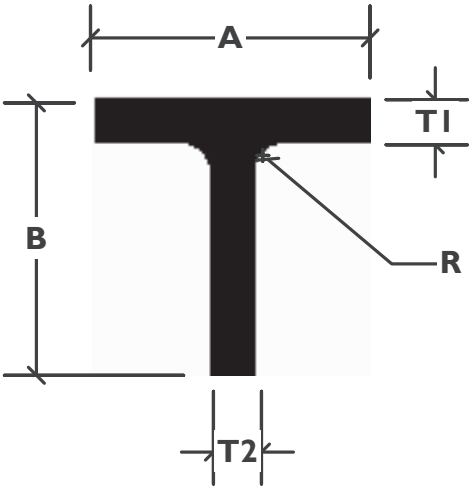
d. Rectangular bar with rounded corners.

A	B	R
3.250	0.375	0.030



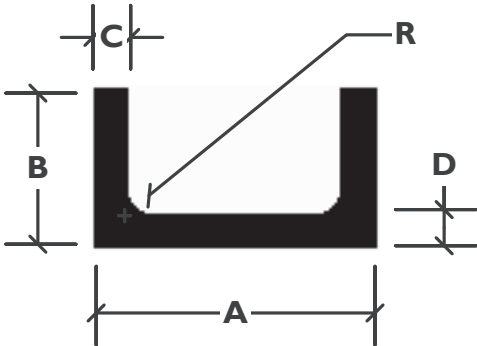
e. Tee.

A	B	T1	T2	R
2.000	2.000	0.125	0.125	0.015

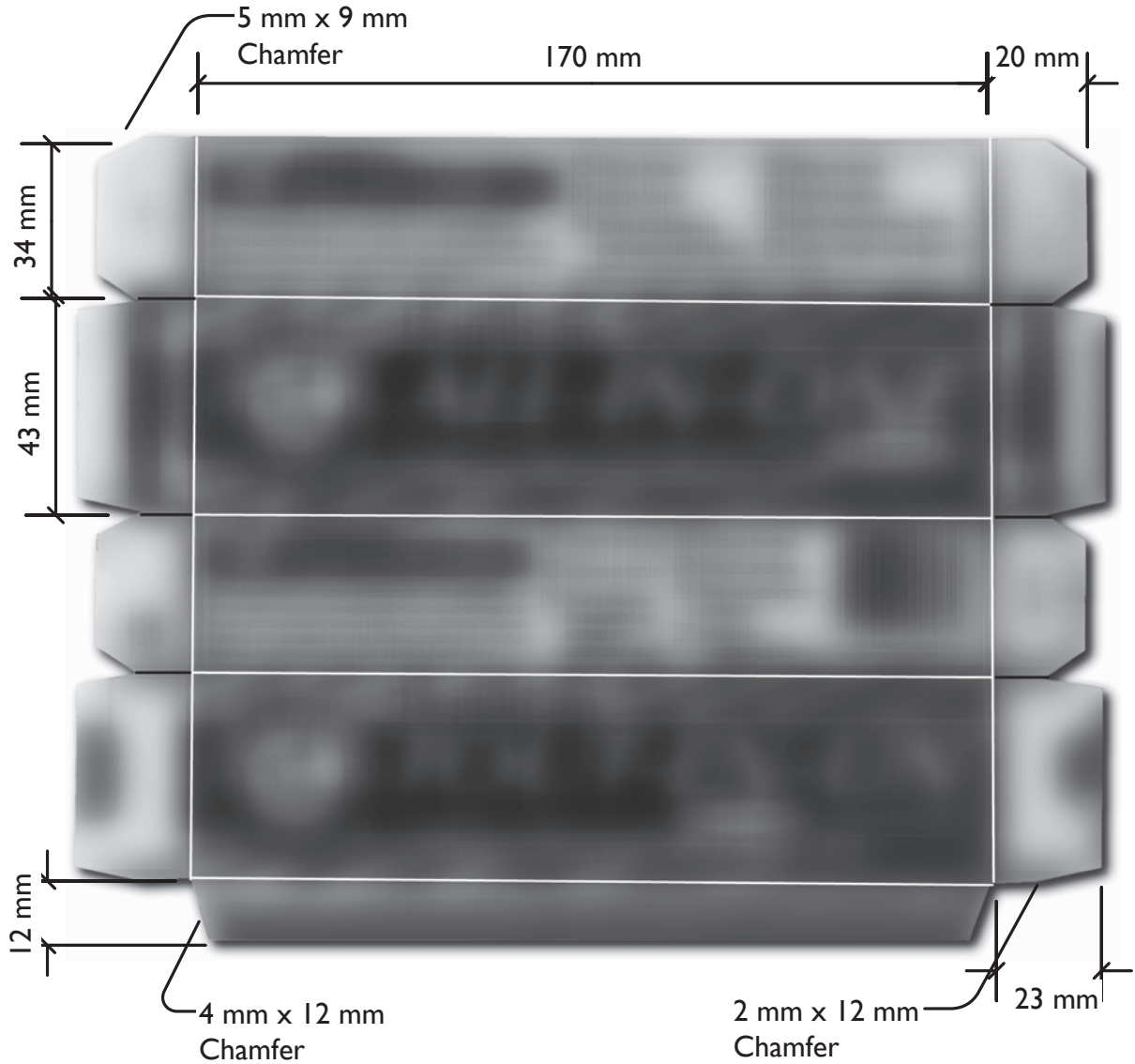


f. Channel with rounded corners.

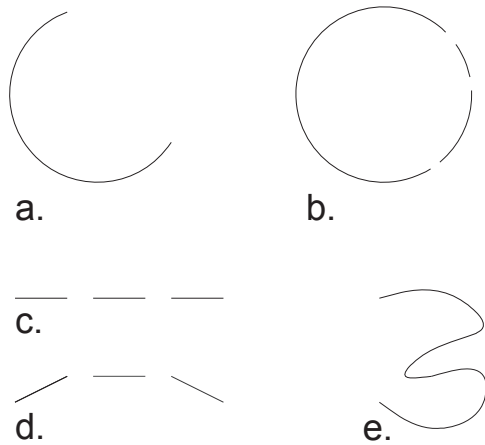
A	B	C	D	R
3.000	1.500	0.375	0.375	0.375



17. Draw the outline and fold lines of the diecut cardboard packaging that holds a tube of toothpaste. Cutlines are shown in black; fold lines in white. All dimensions are in mm. Print the drawing, and then cut and fold along the lines to recreate the cardboard box.



18. From the Companion CD, open the *join.dwg* drawing.



Use the **JOIN** command to perform the following tasks:

- a. Turn the arc into a circle.
- b. Join the three arcs into one circle.
- c. Join the three lines into one line.
- d. Join the three lines into one line.
- e. Join the two splines into one spline.

Which one of the above tasks did not work? Why?

What color did each joined entity become? Why?

CHAPTER REVIEW

1. What two changes can a fillet make to an intersection?
2. When making multiple copies with the **COPY** command, is each copy relative to the first base point entered, or to the point of the last copy made?
3. Can objects of different types be filleted (such as a line and an arc), or must they be alike?
4. What are the three types of arrays?
5. Describe the function of the **MIRROR** command.
6. Which system variable controls whether text is mirrored?
7. What is the *mirror line*?
8. What objects can you use with the **MEASURE** command?
9. How many objects can be offset at one time?
10. Describe the purpose of the **Through** option of the **OFFSET** command?
11. Draw something using the **OFFSET** command's **Through** option.
12. What is the purpose of the **CHAMFER** command?
13. What is the procedure for setting **CHAMFER** distances?
14. Describe the purpose of the **ARRAY** command.
15. Describe a task for which the **REVERSE** command is useful. reversing linetype orientation
16. Can the **ARRAY** command draw arrays at an angle?
17. Under what condition does the **ARRAY** dialog box's **OK** button stay grayed (and unavailable)?
18. To construct an array in the -x direction, you enter a negative value in which option?
19. What could happen to your computer if you construct too large an array?
20. Define *displacement*, as used by the **COPY** command.
21. Which mode helps you copy objects precisely horizontally and vertically?
22. What are the aliases for the following commands:
COPY
MIRROR
OFFSET
FILLET
23. What problem can occur when the **OFFSET** command is applied to curved objects?
24. How does the **OFFSETGAPTYPE** system variable affect polylines?
25. Does the **MEASURE** command measure the lengths of objects?
26. What two kinds of objects does the **DIVIDE** command place along objects?
27. When you specify 5 segments for the **DIVIDE** command, how many objects does it place?
28. Can parallel lines be filleted?
29. Describe the purpose of the **FILLET** and **CHAMFER** commands' **Trim** options.
30. What is the difference between a fillet and a chamfer?
31. Can all vertices of a polyline be chamfered at once?
 If so, how?
32. Describe the purpose of the **REVCLOUD** command.
33. What is another name for "redlines"?
34. How does the **DELOBJ** system variable affect the **Object** option of the **REVCLOUD** command?

35. Explain the purpose of the **MARKUP** command.
36. Describe the role of the Design Review (DWF Composer) software.
37. How are DWF files created?
38. What command converts arcs into circles?
39. Which key do you hold down to make a zero-radius fillet?
A zero-distance chamfer?
40. What is the purpose of the **Undo** option of the **COPY** command?
41. What are *colinear* lines?

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CHAPTER 9

Additional Editing Methods

In previous chapters, you learned numerous methods of creating and changing objects. In this chapter, you learn AutoCAD's other editing commands, including some that may already be familiar to you from grips editing.

This chapter summarizes the remaining editing commands used in two-dimensional drafting:

ERASE deletes objects from drawings, and **OOPS** brings them back.

BREAK removes portions of objects.

TRIM, **EXTEND**, and **LENGTHEN** change the length of open objects.

STRETCH makes portions of objects larger and smaller.

MOVE moves objects in the drawing.

ROTATE rotates objects.

SCALE changes the size of objects.

CHANGE changes the size, properties, and other characteristics of objects.

EXPLODE and **XPLODE** reduce complex objects into their simplest forms.

PEDIT edits polylines.

Grips Editing and **Direct Distance Entry** edit objects directly.

TRACKING and **M2P** assist with direct drawing.

NEW TO AUTOCAD 2010 IN THIS CHAPTER

- **PEDIT** turns splines into polylines.
- **MOVE**, **ROTATE**, and **SCALE** now edit individual segments of polylines.
- **EXPLODE** and **XPLODE** remove constraints from exploded objects.



The **ERASE** command deletes objects from drawings. As an alternative to this command, you can select one or more objects, and press the **DEL** (delete) key.

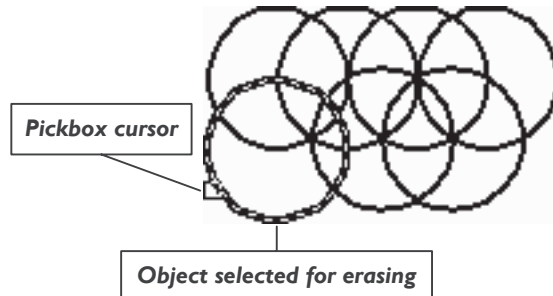
TUTORIAL: ERASING OBJECTS

1. From the CD, open the *erase.dwg* drawing file.
2. To erase one or more objects, start the **ERASE** command:
 - In the ribbons's 2D Home tab, pick the **Erase** button in the Modify panel.
 - At the 'Command:' prompt, enter the **erase** command:
Command: **erase** (Press **ENTER**.)
 - Alternatively, enter the **e** alias at the 'Command:' prompt.

3. In all cases, AutoCAD prompts you to select the objects you want erased:
Select objects: (Pick one or more objects.)
Select objects: (Press **ENTER** to end object selection.)

Notice that the selected objects disappear from the drawing.

At the "Select objects:" prompt, the crosshair cursor changes to the tiny, square pickbox for selecting one object at a time (the *point* method of selection).



If you wish to use another object selection option, such as Window or Crossing, enter the option name at the "Select objects:" prompt. Following erasure, it may be necessary to use the **REDRAW** command to refresh the screen.

RECOVERING ERASED OBJECTS

AutoCAD has two commands for restoring erased objects:

- **OOPS** returns the last erased object(s).
- **U** undoes the last operation, including erasure.

Let's look at each.

Oops

The **OOPS** command restores the objects that were last erased from the drawing.

Command: **oops**

Notice that the erased object(s) are returned to the drawing.

This command also works in conjunction with the **BLOCK** and **-BLOCK** commands, when the original objects were erased in making the block.

U

If you find yourself unable to use **OOPS** to return erased objects, remember that you can also reverse the deletion with the **U** command.



Notes To erase the object you just drew, enter **ERASE** with the **Last** option. A peculiarity is that AutoCAD only erases the last-drawn object if it is visible on the screen.

Similarly, to erase the previous selection set, use **ERASE Previous**. You can erase the entire drawing with the **ERASE All** command. To do the same using keyboard shortcuts, press **CTRL+A** and then **DEL**.



BREAK

The **BREAK** command partially and fully erases objects, as well as cracks them.

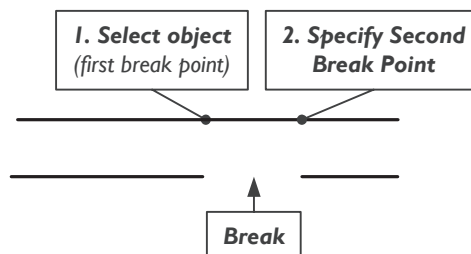
To “break” an object, select two points on it. The portion of the object between the two points is erased. This command removes a portion of almost any object, and shortens open objects — depending on the place you pick. The following objects can be broken: arcs, circles, ellipses, elliptical arcs, lines, polylines, rays, splines, and xlines.

BREAK can remove an object entirely, like the **ERASE** command. Here’s how: for the first point, snap to one end of a line, and then snap to the other endpoint for the second point. The entire line disappears from the drawing. Use **U** to recover, if necessary.

BREAK is useful for “cracking” objects: click the same point twice, and **BREAK** creates two segments that touch. **BREAK** operates on just one object at a time.

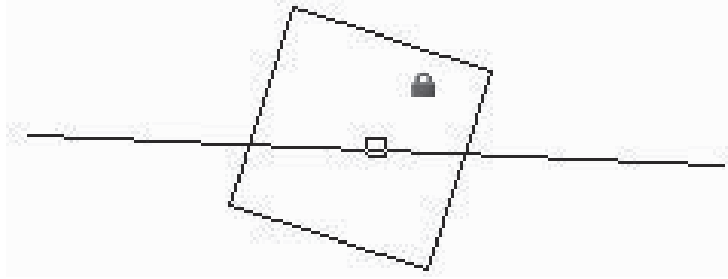
TUTORIAL: BREAKING OBJECTS

- To remove a portion of an object, start the **BREAK** command:
 - In the ribbon’s 2D Home tab, pick the **Break** button in the Modify panel.
 - At the ‘Command:’ prompt, enter the **break** command:
Command: **break** (Press ENTER.)
 - Alternatively, enter the **b** alias at the ‘Command:’ prompt.
- In all cases, AutoCAD prompts you to select the single object you want broken:
Select object: (Pick one object at point 1.)
The point you pick is crucial, because it becomes the first point of the break.
- Pick a second point, the other end of the break:
Specify second break point or [First point]: (Pick point 2.)





Note AutoCAD displays a padlock icon when you attempt to select objects on locked layers. Objects on locked layers can be seen but not edited.



To edit these objects, you must first unlock their layer(s).

BREAKING OBJECTS: ADDITIONAL METHOD

By default, the first point you select on the object becomes the *first* break point. To redefine the first break point, enter **F** in response to the prompt. Select another point to be the first break point.

Redefining the first point is useful when the drawing is crowded, or when the break occurs at an intersection where pointing to the object at the first break point might result in selecting the wrong object.

- **First** defines the first pick point.

Let's look at this option.



First

The **First** option allows you to redefine the first pick point, providing greater control over the segment being broken out.

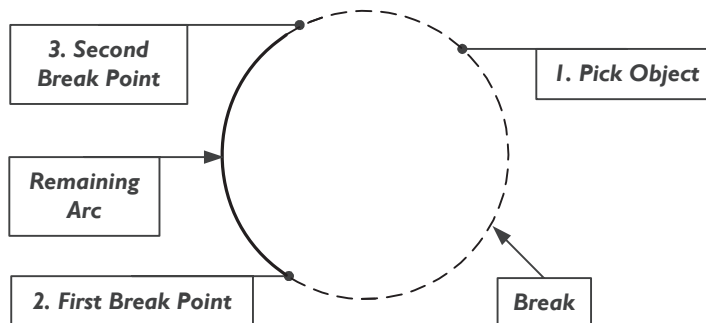
Command: **break**

Select object: *(Pick object.)*

Specify second break point or [First point]: **f**

Specify first break point: *(Pick point 1.)*

Specify second break point: *(Pick point 2.)*



Note The first tip I ever learned from technical editor Bill Fane was that sometimes it is easier to create an arc by drawing a circle, and then using the **BREAK** command to remove the unwanted portion.

The BREAK command affects objects in different ways:

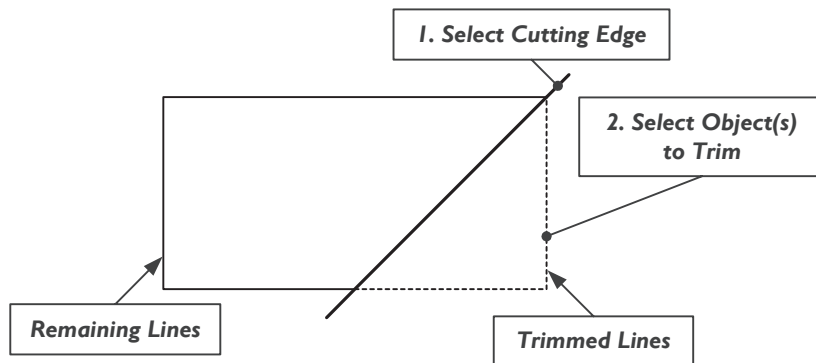
Object	Break Action
Arcs, Lines	Removes portion of line or arc between the pick points. When one point is on the line, but the other point is off the end of the line, the line is “trimmed back” to the first break point.
Circles	Breaks into an arc. Unwanted piece is determined by going counterclockwise from the first point to the second point.
Ellipses	Breaks into an elliptical arc in a manner like circles.
Splines	Reacts in the same manner as lines and arcs.
Traces	Reacts in the same manner as lines, except that the new endpoints are trimmed square.
Polylines	Cuts wide polylines squarely, as with traces. Breaking closed polylines creates open polylines.
Viewports	Does not brake.
Xlines	Breaks into rays.

TRIM, **EXTEND**, AND **LENGTHEN**

The TRIM, EXTEND, and LENGTHEN commands change the length of open objects.

The TRIM command shortens objects by defining other objects as *cutting edges*; any portion of the object beyond the cutting line is “cut off.”

AutoCAD needs to know two pieces of information for trimming: (1) the object(s) to be used as the cutting edges, and (2) the portion of the object to be removed.



Cutting edges are any combination of arcs, circles, ellipses, lines, viewports, rays, regions, splines, text, xlines, hatches, 2D and 3D polylines. Polylines with nonzero width trim to the centerline of the polyline.

A quick method is to select *all* objects as cutting edges: at the ‘Select objects <select all>.’ prompt, press ENTER, and AutoCAD selects all objects in the drawing, including those not in the current view. A previously undocumented method of quickly selecting many objects to trim is the **Fence** selection mode. The part that is picked is trimmed.

TUTORIAL: SHORTENING OBJECTS

- I. To shorten one or more objects, start the **TRIM** command:
 - In the ribbon’s 2D Home tab, pick the **Trim** button in the Modify panel.

- At the 'Command:' prompt, enter the **trim** command:
Command: **trim** (Press ENTER.)
 - Alternatively, enter the **tr** alias at the 'Command:' prompt.
2. In all cases, AutoCAD reports the current trim settings, and then prompts you to select cutting edges:
Current settings: Projection=UCS Edge=None
Select cutting edges ...
Select objects or <select all>: (Pick one or more objects as cutting edges, or press ENTER to select all objects in the drawing.)
Select objects: (Press ENTER to end object selection.)
 3. Select the object to be trimmed. *Warning!* AutoCAD trims the portion you select.
Select object to trim or shift-select to extend or
[Fence/Crossing/Project/Edge/eRase/Undo]: (Pick objects to trim, or enter an option.)
 4. AutoCAD repeatedly prompts you to select additional objects to trim. When done trimming, press **ENTER**:
Select object to trim or shift-select to extend or
[Fence/Crossing/Project/Edge/eRase/Undo]: (Press ENTER to exit.)

When you select objects that cannot serve as cutting edges, such as blocks and dimensions, AutoCAD displays the message:

No edges selected.

If an object cannot be trimmed, AutoCAD displays:

Cannot TRIM this object

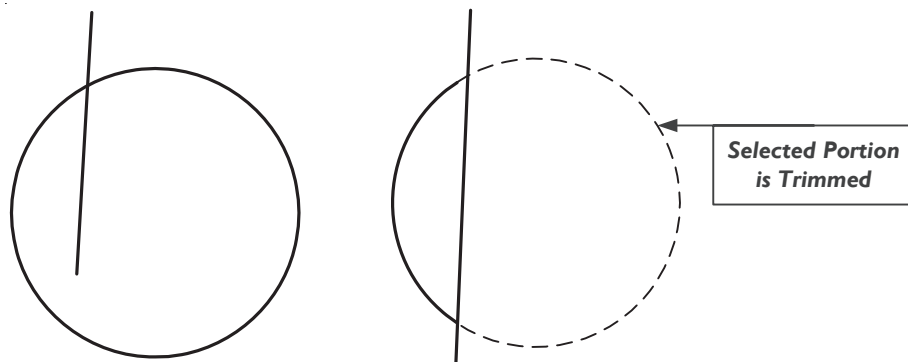
If the object to be trimmed does not intersect a cutting edge, AutoCAD complains:

Object does not intersect an edge.

Trimming Closed Objects

To be trimmed, closed objects, such as circles and polygons, must be intersected at least twice by the cutting edge, such as by a line drawn through two points on a circle's circumference. If only one cutting edge intersects the closed object, AutoCAD complains:

Circle must intersect twice.



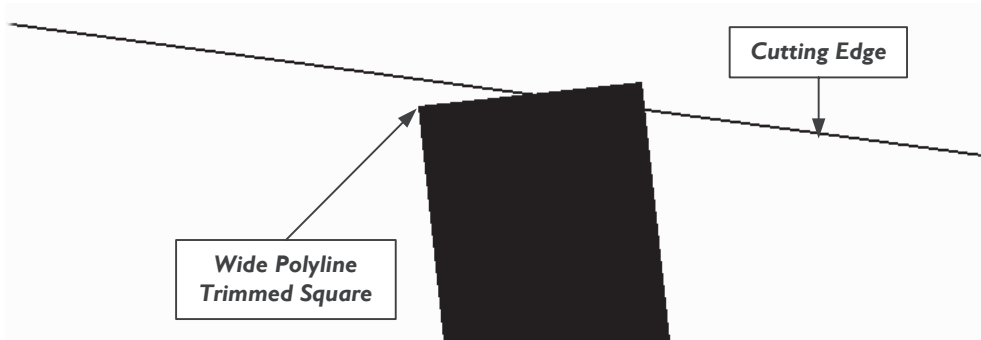
Left: Circle cannot be trimmed, because the line intersects it at only one location.

Right: This circle can be trimmed, because the line intersects it twice.

You can use two cutting edges (two lines) that cross the circle.

Trimming Polylines

Polylines are trimmed at the intersection of the center line of the polyline and the cutting edge. The trim is a square edge. Therefore, if the cutting edge intersects a polyline of nonzero width at an angle, the square-edged end may protrude beyond the cutting edge.



Notes The **TRIM** and **EXTEND** commands can switch roles:

To *extend* an object during the **TRIM** command, hold down the **SHIFT** key while selecting the object. Similarly, to *trim* an object during the **EXTEND** command, hold down the **SHIFT** key while selecting the object.

Hatch patterns can be cutting edges or the objects being trimmed. In other words, the **TRIM** command can trim hatch and fill patterns, and use them to trim other objects. (The **EXTEND** and **LENGTHEN** commands have no effect on hatches.)

TRIMMING OBJECTS: ADDITIONAL METHODS

The **TRIM** command has several options:

- **Select all** selects all objects as cutting edges.
- **Fence** and **Crossing** select objects to be trimmed.
- **Project** determines trimming in 3D space.
- **Edge** toggles actual and implied cutting edges in 3D space.
- **eRase** erases objects.
- **Undo** undoes the last trim.
- **SHIFT-select** extends the object to the cutting line, instead of trimming.

Let's look at each option.

Select All

The **Select all** option selects all objects in the drawing as cutting edges — at least those not on frozen layers. When you press ENTER at the 'Select objects or <select all>:' prompt, AutoCAD selects all objects, and then stops repeating the 'Select objects:' prompt.

Fence and Crossing

The **Fence** and **Crossing** options select objects to be trimmed. Their inclusion in the prompt may be puzzling to you: can't any selection mode be used? No. For some peculiar reason, the original **TRIM** command allowed you only to pick objects singly for trimming.

The **Fence** option prompts:

Specify first fence point: *(Pick a point.)*

Specify next fence point or [Undo]: *(Pick another point.)*

And continues repeating the prompt until you press ENTER. You can back up along the fence path by entering the **Undo** option.

Using the **Crossing** option can be ambiguous; Autodesk notes that AutoCAD follows the rectangular crossing window clockwise from the first pick point. The **Crossing** option prompts:

Specify first corner: *(Pick a point.)*

Specify opposite corner: *(Pick another point.)*

The crossing window must cross objects; otherwise nothing is trimmed. When you pick an object at the 'Specify first corner:' prompt, AutoCAD abandons **Crossing** mode, and selects the object instead.

Project

The **Project** option is used in 3D drafting to specify the projection AutoCAD uses when it trims objects.

Enter a projection option [None/Ucs/View]: *(Enter an option.)*

The sub-options are:

None specifies no projection; AutoCAD trims only objects that actually intersect with cutting edges.

UCS projects the objects onto the x, y-plane of the current UCS (user-defined coordinate system). AutoCAD trims objects, even if they do not intersect with the cutting edge in 3D space—in effect, flattening 3D objects onto the 2D plane.

View specifies a projection along the current view direction. AutoCAD trims objects that look as if they should be trimmed from your viewpoint, even if they don't physically intersect.

Edge

The **Edge** option is also used in 3D drafting, and determines whether objects are trimmed at an actual cutting edge (objects crossing physically), or at an *implied* cutting edge (objects appearing to cross):

Enter an implied edge extension mode [Extend/No extend]: *(Enter an option.)*

The options are:

Extend projects the cutting edge so that it intersects the object(s).

No extend trims objects at cutting edges that physically intersect; cutting edges are not extended.

eRase

The **eRase** option erases objects, just like the **ERASE** command itself. It is a convenient way of removing whole objects that don't require trimming.

Undo

The **Undo** option undoes the last trim. This allows you to undo trim errors without leaving the command.

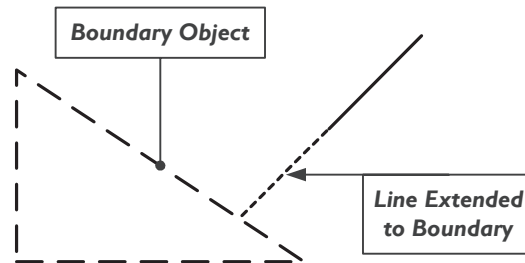
Extend (SHIFT-select)

When you hold down the **SHIFT** key at the "Select object to trim or shift-select to extend" prompt, AutoCAD extends the object to the cutting line, instead of trimming — in effect, reversing the **TRIM** command. The complementary option is available in the **EXTEND** command.

EXTEND

The **EXTEND** command extends objects in a drawing to meet a boundary object. It functions very much like the **TRIM** command. Instead of a cutting edge, this command uses *boundary* objects — the boundary to which selected objects are extended.

AutoCAD needs to know two things: (1) the object(s) to be used as boundary objects, and (2) the object to extend.



Boundary objects can be lines, arcs, circles, polylines, ellipses, splines, rays, xlines, regions, blocks, and viewports (in paper space). If several boundary edges are selected, the objects are extended to the first boundary encountered. If none can be reached, AutoCAD complains:

No edges selected.

Generally, closed objects cannot be extended, specifically text, splines, 3D solids, xlines, mutlilines, ellipses, donuts, hatches, revelclouds, regions, 3D surfaces, and points. (Dimensions can be extended.) If the object cannot be extended, AutoCAD displays:

Cannot EXTEND this object.

Take care! You must point close to the end of the object to extend.

TUTORIAL: EXTENDING OBJECTS

1. To extend one or more objects, start the **EXTEND** command:
 - In the ribbons's 2D Home tab, pick the **Extend** button in the Modify panel.
 - At the 'Command:' prompt, enter the **extend** command:

Command: **extend** (Press ENTER.)
 - Alternatively, enter the **ex** alias at the 'Command:' prompt.
2. In all cases, AutoCAD reports the current extend settings, and then prompts you to select boundary edges:

Current settings: Projection=UCS Edge=None

Select boundary edges ...

Select objects or <select all>: (Pick one or more objects as boundaries.)

Select objects: (Press ENTER to end object selection.)
3. Select the object to be extended. AutoCAD extends the end that you select.

Select object to extend or shift-select to trim or [Fence/Crossing/Project/Edge/Undo]:

(Pick objects to be extended.)
4. AutoCAD repeatedly prompts you to select additional objects to extend. When done extending, press **ENTER**:

Select object to extend or shift-select to trim or [Fence/Crossing/Project/Edge/Undo]:

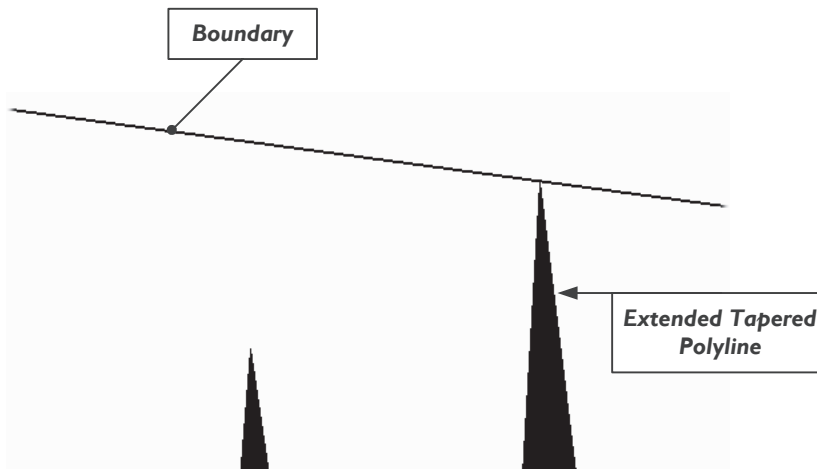
(Press ENTER to exit command.)

Extending Polylines

Polylines of nonzero width are extended until the centerline meets the boundary object; similarly, objects extend to the centerline of the polylines. Only open polylines can be extended. If you attempt to extend a closed polyline, AutoCAD complains:

Cannot extend a closed polyline.

When wide polylines and the boundary intersect at an angle, a portion of the square end of the polyline may protrude over the boundary. Extending tapered polylines adjusts the length of the segments that taper — the taper extends over the longer length.



A tapered polyline extended to a boundary, its taper extended as well.

EXTEND OPTIONS

The **EXTEND** command's options are identical to those of **TRIM**'s.

LENGTHEN

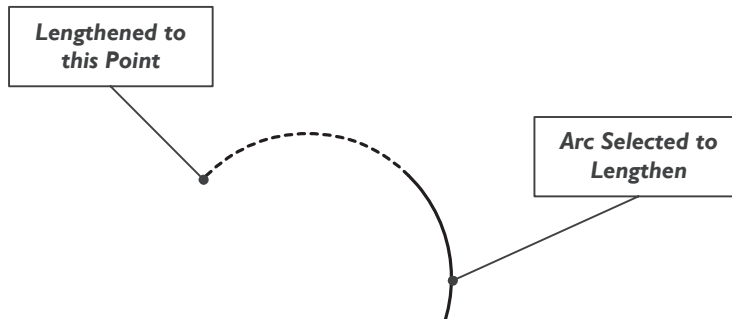
The **LENGTHEN** command is a faster version of **TRIM** and **EXTEND**: it changes the length of open objects, making them longer or shorter; it does not work with closed objects. **LENGTHEN** is faster, because you do not need cutting edges or boundaries: you just point anywhere in the drawing for the change to occur, or else specify a percentage change numerically.

This command works with lines, arcs, open polylines, elliptical arcs, and open splines.

TUTORIAL: LENGTHENING OBJECTS

- To change the length of an open object, start the **LENGTHEN** command:
 - In the ribbons's 2D Home tab, pick the **Lengthen** button in the Modify panel.
 - At the 'Command:' prompt, enter the **lengthen** command:
Command: **lengthen** (Press ENTER.)
 - Alternatively, enter the **len** alias at the 'Command:' prompt.
- In all cases, AutoCAD prompts you to select an object. AutoCAD reports the length of the object:
Select an object or [DElta/Percent/Total/DYnamic]: (Select one open object.)
Current length: 24.6278

3. AutoCAD repeats the prompt. It wants you to select an option:
Select an object or [DElta/Percent/Total/DYnamic]: (Enter an option, such as **dy**.)
4. Curiously, AutoCAD has forgotten which object you selected; you need to select it a second time — or, you can select a different object:
Select an object to change or [Undo]: (Select one open object.)
5. As you move the cursor, notice that AutoCAD ghosts in the lengthened line. When satisfied with the new length, click.
Specify new end point: (Move the cursor, and then click.)
6. AutoCAD repeats the prompt. When done lengthening, press **ENTER**:
Select an object to change or [Undo]: (Press **ENTER** to exit the command.)



Note When you select an object at the initial “Select an object:” prompt, AutoCAD reports on its length, as follows: *Current length: 10.2580*. For an arc, AutoCAD also reports its angle:

Current length: 8.4353, included angle: 192.

When you select a closed object, such as a circle or closed polyline, AutoCAD complains, “This object has no length definition.” You cannot lengthen dimensions, hatch patterns, and splines.

LENGTHENING OBJECTS: ADDITIONAL METHODS

The **LENGTHEN** command changes the length of open objects by four methods:

Delta — changes by an incremental amount.

Percent — changes by a percentage.

Total — changes to the total amount.

DYnamic — changes by cursor movement.

Let’s look at how each option.

Delta

The **DELta** option changes the length by adding the indicated amount to the object. Enter a negative number to shorten the object. As an alternative to typing a value, you can indicate the amount by picking two points anywhere in the drawing.

Command: **lengthen**

Select an object or [DElta/Percent/Total/DYnamic]: **de**

Enter delta length or [Angle] <0.0000>: (Enter a number, or pick two points on screen.)

Select an object to change or [Undo]: (Select object.)

Select an object to change or [Undo]: (Press **esc** to exit the command.)

Note that this command lengthens the open object at the end you select.

The **Angle** option changes the angle of a selected arc or polyarc. The command lengthens the end you select:

Enter delta angle <0>: *(Type an angle, or pick two points.)*

Select an object to change or [Undo]: *(Select the arc.)*

Select an object to change or [Undo]: *(Press esc to exit the command.)*

The arc is lengthened by the angle you specify. When you select two points, AutoCAD uses the angle of the rubber-band line, not the length of the line.

The angle you specify must be small enough for the arc not to total 360 degrees. If the angle is too large, AutoCAD curtly informs you: "Invalid angle."

Percent

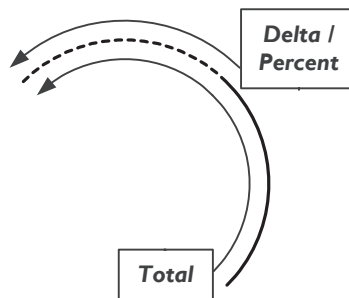
The **Percent** option changes the length by a percentage of the object. For example, entering **25** shortens a line to 25 percent of its original length, while entering **200** doubles its length.

Select an object or [DElta/Percent/Total/DYnamic]: **p**

Enter percentage length <100.0000>: *(Enter a percentage, such as 25.)*

Select an object to change or [Undo]: *(Select the object.)*

Select an object to change or [Undo]: *(Press esc to exit the command.)*



Total

The **Total** option changes the length of a line by an absolute length. For example, a value of **5** changes the line to a length of 5.0 units, no matter its existing length.

Select an object or [DElta/Percent/Total/DYnamic]: **t**

Specify total length or [Angle] <1.0000>: *(Enter a value, such as 5.)*

Select an object to change or [Undo]: *(Select the object.)*

Select an object to change or [Undo]: *(Press esc to exit the command.)*

This option changes arcs to an absolute length. The **Angle** option changes the length of an arc to the included specified angle.

Dynamic

The **DYnamic** option visually changes the length of open objects. Notice that this option reverses the order: (1) first you select the object to lengthen, and (2) then you specify the length.

Select an object or [DElta/Percent/Total/DYnamic]: **dy**

Select an object to change or [Undo]: *(Select the object.)*

Specify new end point: *(Pick a point.)*

Select an object to change or [Undo]: *(Press esc to exit the command.)*

After you select the object, the length of the object changes as you move the cursor. You can use object snap modes to make the dynamic lengthening more accurate.

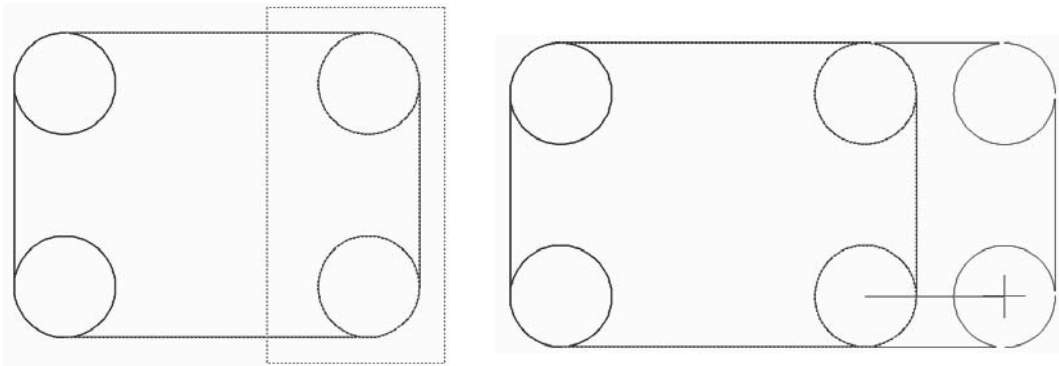
STRETCH

The **STRETCH** command makes portions of objects larger and smaller, *and* moves selected objects, while retaining their connections to other objects. As you will see, it is one of the most useful editing commands in AutoCAD — a distant second only to the **UNDO** command. The command works with lines, arcs, elliptical arcs, solids, traces, rays, splines, and polylines.

STRETCHING — THE RULES

You must understand the rules associated with the **STRETCH** command to execute it properly:

Rule 1: Crossing Selection. You can use any object selection mode, but at least one must be **Crossing** or **CPolygon**. If you do not use a *windowed* selection, AutoCAD complains, “You must select a crossing or polygon window to stretch.” The command is no longer restricted to using the last window specified; instead, **STRETCH** now remembers all the objects selected by multiple **Crossing** selection rectangles. The window must include at least one vertex or endpoint of the object.



Objects within the selection window being moved; those crossing the window being stretched.

Rule 2: Move vs. Stretch. If the object is completely inside the selection, it will be moved rather than stretched. Objects — specifically arcs, elliptical arcs, lines, polyline segments, 2D solids, rays, traces, and splines — entirely within the selection window are moved as with the **MOVE** command. The endpoints within the selection window are moved, while those outside remain fixed.

Arcs are stretched like lines, except that the arc’s center, start, and endpoints are adjusted so the distance from the midpoint of the chord to the arc is constant. You can stretch either end of the polyline, and move its vertices. Spline-curve and fit-curve polylines can be stretched if the crossing window includes the original vertex (even though it is invisible). When tangent-arc polyline segments are stretched, they lose tangency.

This command does not stretch 3D solids or text.

Rule 3: Definition Points. Some objects, such as circles and blocks, cannot be stretched; they are either moved or left alone by the **STRETCH** command, depending on their *definition point*. If this point

lies inside the selection window, the object is moved; if outside, it is not affected.

Object	Definition Point
Point	Center of the point.
Circle	Center point of the circle.
Block	Insertion point.
Text	Insertion point of the text line.

TUTORIAL: STRETCHING OBJECTS

1. To stretch one or more objects, start the **STRETCH** command:
 - In the ribbons’s 2D Home tab, pick the **Stretch** button in the Modify panel.
 - At the ‘Command:’ prompt, enter the **stretch** command:
Command: **stretch**
 - Alternatively, enter the **s** alias at the ‘Command:’ prompt.
2. In all cases, AutoCAD prompts you to select the objects using a crossing selection mode. If you do not enter a selection option, such as **Crossing**, AutoCAD defaults to **AU**tomatic.
Select objects to stretch by crossing-window or crossing-polygon...
Select objects: **c**
Specify first corner: *(Pick a point.)*
Specify opposite corner: *(Pick another point.)*
Select objects: *(Press ENTER to end object selection.)*
3. Identify the points that measure the displacement:
Specify base point or Displacement <Displacement>: *(Pick a point, using an object snap mode if necessary.)*
Specify second point of displacement: *(Pick the destination point.)*

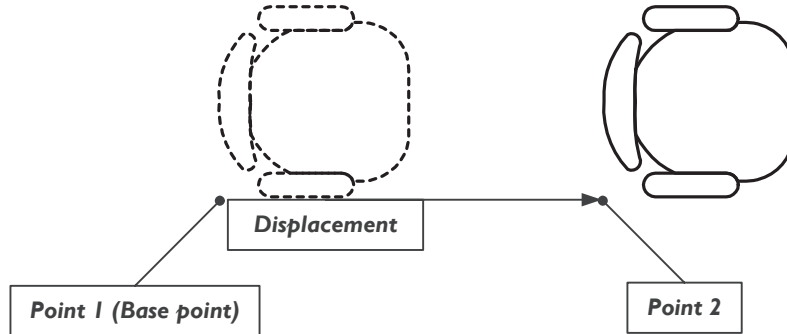
For the **Displacement** option, you can enter Cartesian, polar, cylindrical, or spherical coordinates — without the @ prefix, because AutoCAD assumes relative coordinates. The next time you use the **STRETCH** command, enter the **Displacement** option to recall the previous displacement distance.

An alternative to the **STRETCH** command is grips editing, as described later.



The **MOVE** command moves objects in the drawing.

It is similar to the **COPY** command in that AutoCAD needs to know two things: (1) the objects to move, and (2) the displacement — the distance to move. (See Chapter 8, “Constructing Objects.”)



TUTORIAL: MOVING OBJECTS

1. To move one or more objects, start the **MOVE** command:
 - In the ribbons's 2D Home tab, pick the **Move** button in the Modify panel.
 - At the 'Command:' prompt, enter the **move** command:
Command: **move** (Press ENTER.)
 - Alternatively, enter the **m** alias at the 'Command:' prompt.
2. In all cases, AutoCAD prompts you to select the objects to move:
Select objects: (Select one or more objects.)
Select objects: (Press ENTER to end object selection.)
3. Identify the point from which the displacement is measured:
Specify base point or Displacement <Displacement>: (Pick point 1.)
4. And pick the second point:
Specify second point of displacement or <use first point as displacement>: (Pick point 1.)

The first point need not be on the object to be moved; using a corner point or another convenient point of reference on the object makes the displacement easier to visualize. I find that object snap and ortho mode help make the move more precise.

The next time you use the **MOVE** command, select the **Displacement** option to recall the previous displacement distance.

An alternative to the **MOVE** command is grips editing, as described in Chapter 10. The **GTDEFAULT** system variable determines whether the **MOVE** or **MOVE3D** command is activated when entering “move” in 3D views; the same holds for **ROTATE** and **ROTATE3D**.

Command-less Moving

You can move objects in AutoCAD without using any commands. Here's how:

1. Select an object. Notice the blue grips.
2. Grab the object away from the grips by holding down the left mouse button.
3. Wait.



4. After a second or two, the move cursor appears. You can now drag the object in the drawing, effectively moving it.



ROTATE

The **ROTATE** command rotates objects about a base point. (The *base point* is the point about which selected objects rotate.)

You can specify the rotation angle three ways: by entering an angle, by dragging the angle, or by choosing a reference angle. (The similar sounding **REVOLVE** command creates 3D solid models.)

TUTORIAL: ROTATING OBJECTS

1. To turn one or more objects, start the **ROTATE** command:
 - In the ribbons's 2D Home tab, pick the **Rotate** button in the Modify panel.
 - At the 'Command:' prompt, enter the **rotate** command:

Command: **rotate** (Press ENTER.)
 - Alternatively, enter the **ro** alias at the 'Command:' prompt.
2. In all cases, AutoCAD indicates the current rotation settings, and then prompts you to select the objects you want rotated:

Current positive angle in UCS: ANGDIR=counterclockwise ANGBASE=0

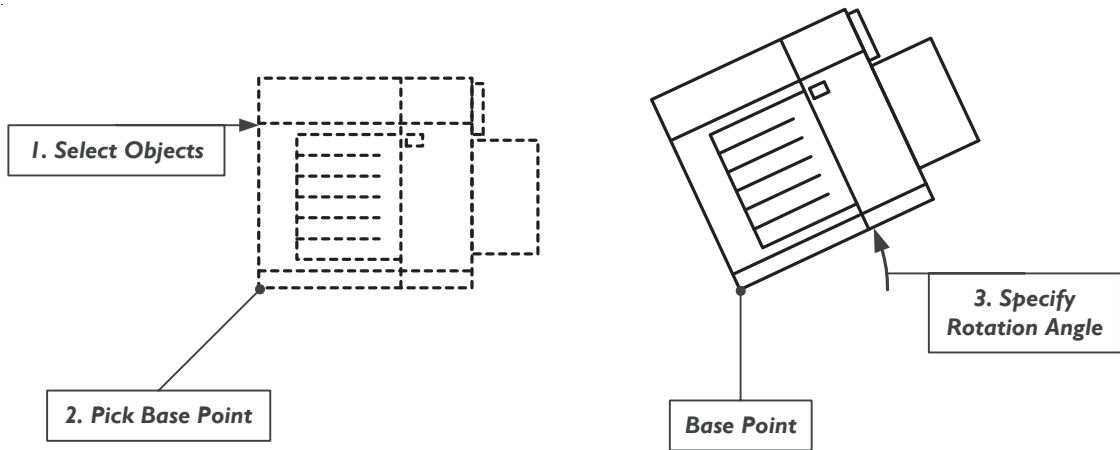
Select objects: (Pick one or more objects.)

Select objects: (Press ENTER to end object selection.)
3. Identify the point about which to rotate the objects:

Specify base point: (Pick a point, such as the lower left corner.)

The base point need not be on the object, but can be anywhere in the drawing.
4. Specify the angle of rotation:

Specify rotation angle or [Copy/Reference] <90>: (Enter an angle, such as **45**.)



You can specify a simple angle, or change one angle to another. For example, if an object is currently oriented at 58 degrees, and you wish to rotate the object to 26 degrees, rotate the object by the difference of -32 degrees. Positive angles rotate counterclockwise; negative angles, clockwise.

ROTATING OBJECTS: ADDITIONAL METHODS

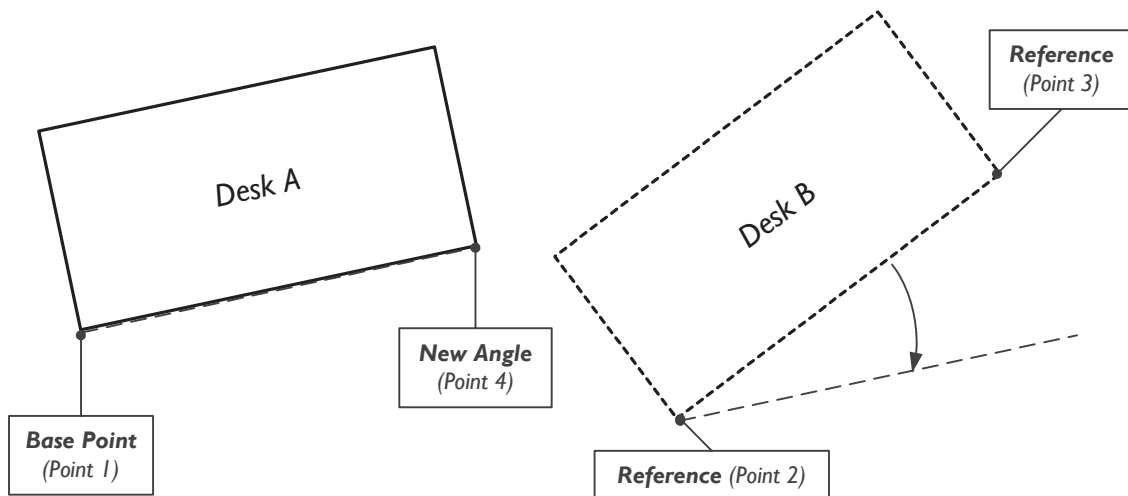
The ROTATE command has alternative options for specifying the angle:

- **Reference** option rotates relative to another angle.
- **Copy** option rotates a copy of the object.
- **Dragging** option rotates the object in real-time.

Let's look at each.

Reference

The **Reference** option aligns the object to another. You don't need to know the angle of the other object, but object snaps are most useful. In this example, we want Desk B to be at the same slant as Desk A.



Specify base point: **end**
 of (Pick point 1.)
 Specify rotation angle or [Reference]: **r**
 Specify the reference angle <0>: **end**
 of (Pick point 2.)

Second point: : **end**
 of *(Pick point 3.)*
 Specify the new angle: **end**
 of *(Pick point 4.)*

Copy

The **Copy** option rotates a copy of the object, leaving the original in place. When you enter **c** at the ‘Specify rotation angle or [Copy/Reference] <90>:’ prompt, AutoCAD reports:

Rotating a copy of the selected objects.

The option is temporary. The next time you use the **ROTATE** command, you have to reenter the **Copy** option.

Dragging

To rotate the objects in real time, move the cursor in response to the “Specify rotation angle or [Reference]:” prompt. Keep an eye on the status line as it reports the x, y, z coordinates and angle. Click to fix the object at the location shown on the screen.



SCALE

The **SCALE** command changes the size of objects. Contrary to what the name suggests, this command does not change the *scale* of objects in the drawing — but their *size*. The x, y, and z directions of the objects are changed equally.

AutoCAD needs to know three things: (1) the objects to be resized, (2) the base point from which the objects are resized, and (3) the amount to resize.

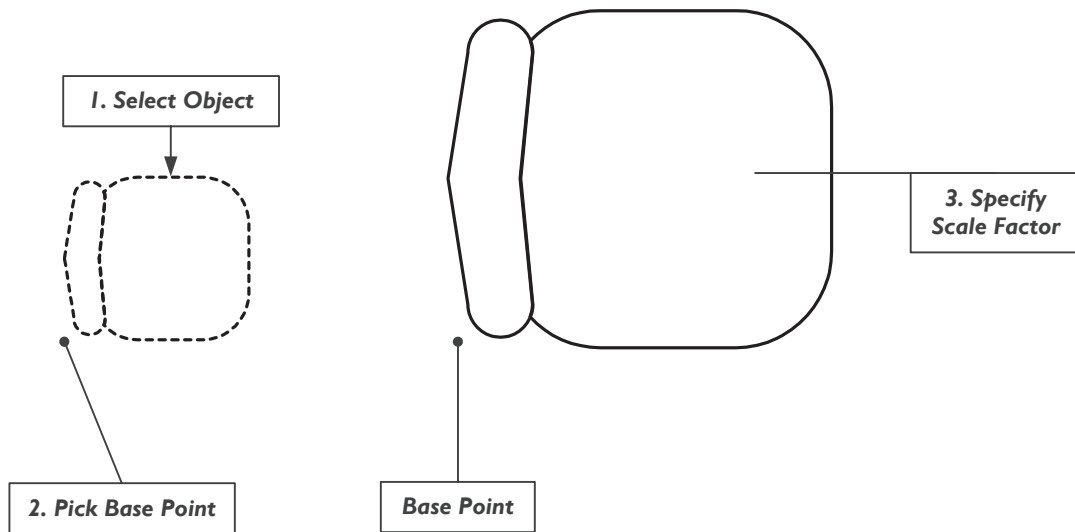
TUTORIAL: CHANGING SIZE

1. To resize one or more objects, start the **SCALE** command:
 - In the ribbons’s 2D Home tab, pick the **Scale** button in the Modify panel.
 - At the ‘Command:’ prompt, enter the **scale** command:
 Command: **scale** *(Press ENTER.)*
 - Alternatively, enter the **sc** alias at the ‘Command:’ prompt.
2. In all cases, AutoCAD prompts you to select the objects you want resized:

Select objects: *(Pick one or more objects.)*

Select objects: *(Press ENTER to end object selection.)*
3. Identify the point about which the objects are rotated:

Specify base point: *(Pick a point, such as the lower left corner.)*



The base point can be located anywhere in the drawing. This point remains stationary, and the object is resized from that point.

4. Specify the resize factor:

Specify scale factor or [Copy/Reference] <1.0000>: *(Enter a factor, such as 2 or 0.5.)*

Entering a decimal factor makes objects smaller: a factor of 0.25 makes an object 25 percent of the original size. Factors larger than 1.0 increase the size of the objects. For example, entering 2.0 makes the object twice the original size in the direction of all three axes.

Instead of entering the scale factor, you can specify the size by dragging the cursor or by referencing a known length, and then entering a new length.

CHANGING SIZE: ADDITIONAL METHODS

The SCALE command has two options:

- **Copy** rotates a copy of the objects.
- **Reference** changes the size relative to other distances.

Let's look at them.

Copy

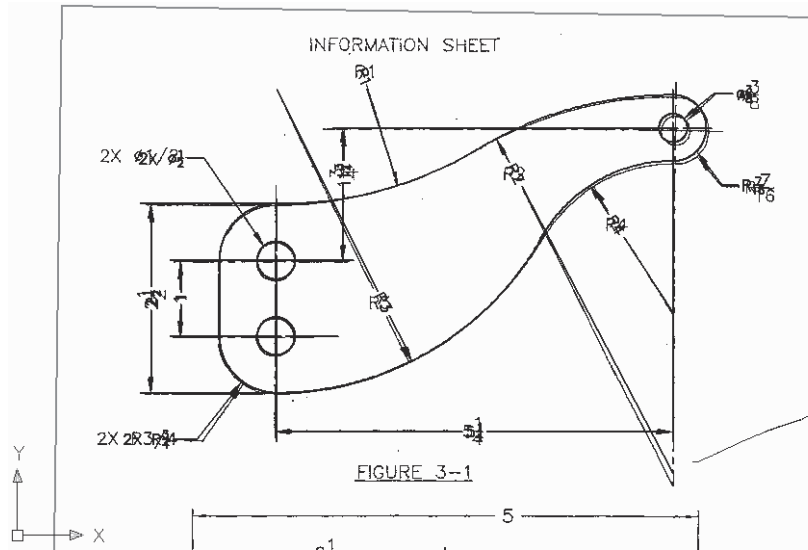
The **Copy** option scales a copy of the objects, leaving the original in place. When you enter **c** at the 'Specify scale factor or [Copy/Reference] <1.0>:' prompt, AutoCAD reports:

Scaling a copy of the selected objects.

The option is temporary. The next time you use the SCALE command, you have to reenter the **Copy** option.

Reference

The **Reference** option adjusts objects to a “correct” size. This is particularly useful when you scan a drawing, and then bring the raster image into AutoCAD with the **IMAGE** command. To trace over the scanned image, you first have to change the image to the right size: you achieve this with the **Reference** option. All you need is the true length of just one line in the scanned image.



The **Reference** option of the **SCALE** and **ROTATE** commands correcting the size and skew of scanned images.

1. At the "Select objects:" prompt, select the raster image:
Command: **scale**
Select objects: *(Pick the edge of the raster image.)*
Select objects: *(Press ENTER.)*
2. For the base point, select the lower left corner of the raster image.
Specify base point: *(Pick.)*
3. Specify the **Reference** option:
Specify scale factor or [Copy/Reference] <1.0>: **r**
4. Select the end of the known line in the raster image.
Specify reference length <1>: *(Pick one end of the raster line.)*
5. Select the other end of the raster line; this allows AutoCAD to measure the distance of the line:
Specify second point: *(Pick other end of raster line.)*
6. For the new length, enter the value of the known dimension.
Specify new length: *(Enter value.)*

AutoCAD resizes the scanned image, making it the correct size. Use the **ROTATE** command to correct a skewed image. (If the scanned image is warped, you need to correct it with “rubber sheeting,” a feature not available in AutoCAD.) To change the aspect ratio of an image, convert it to a block, and then insert it with differing x and y scale factors; explode the block, and the image maintains its new aspect ratio.

The **SCALE** command is also useful for converting an Imperial drawing to metric, and vice versa.

CHANGE

The **CHANGE** command changes the size, properties, and other characteristic of objects. It is similar to, but has more capabilities than, **CHPROP**, but **PROPERTIES** is the easiest and most versatile of all. (See Chapter 7.)

TUTORIAL: CHANGING OBJECTS

1. To change one or more objects, start the **CHANGE** command:
 - At the 'Command:' prompt, enter the **change** command:
Command: **change** (Press ENTER.)
 - Alternatively, enter the **-ch** alias at the 'Command:' prompt.
2. In all cases, AutoCAD prompts you to select the complex objects you want to change:

Select objects: (Pick one or more objects.)

Select objects: (Press ENTER to end object selection.)
3. The **Properties** options are somewhat different from the **CHPROP** command:

Specify change point or [Properties]: (Pick a point.)

The **CHANGE** command's **Elev** option changes the elevation of objects, an option missing from **CHPROP**. Although the Properties palette is generally handier, **CHANGE** offers this advantage: the **Change Point** option reacts differently depending on the object selected, as described next.

CHANGING OBJECTS: ADDITIONAL METHODS

Here's how the **CHANGE** command's **Change Point** option affects different objects:

- **Lines** change endpoints.
- **Circles** change radius.
- **Blocks** change insertion point and rotation angle.
- **Text** changes properties.
- **Attribute Text** changes properties.

Circles

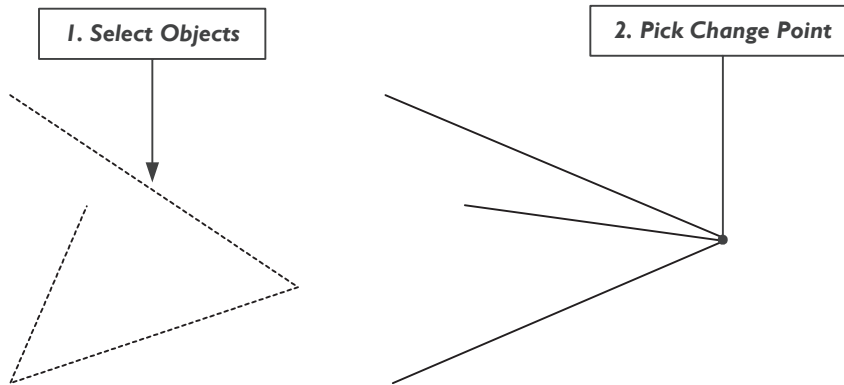
The **Change Point** option changes the circle's radius. When more than one circle is selected, AutoCAD repeats the prompt for each circle:

Specify new circle radius <no change>: (Enter a new radius, or press ENTER.)

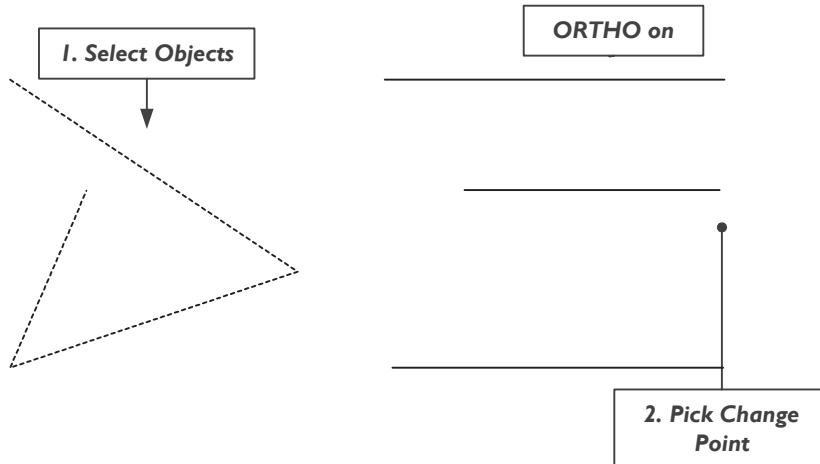
Press ENTER to keep the circle's radius.

Lines

The **Change Point** option moves the endpoints of lines closest to the pick point.



When ortho mode is on, AutoCAD makes the lines parallel to either the x or y axis, depending on which is closest. This is a quick way to straighten out a bunch of crooked lines, and is not available in any other AutoCAD command.



Blocks

The **Change Point** option changes the location and rotation of blocks.

Specify new block insertion point: *(Pick a new insertion point, or press ENTER.)*

Specify new block rotation angle <current>: *(Enter a new angle, or press ENTER.)*

Press ENTER to keep each option in place.

Text

The **Change Point** option changes the position of the text, as well as the text's other properties. (Until the PROPERTIES command's predecessor, DDMODIFY, came along, this was the only way to change text properties in a drawing.)

Specify new text insertion point <no change>: *(Pick a new insertion point, or press ENTER.)*

Enter new text style <current>: *(Enter a different text style, or press ENTER.)*

Specify new height <current>: *(Enter a new height, or press ENTER.)*

Specify new rotation angle <current>: *(Enter a new angle, or press ENTER.)*

Enter new text <current>: *(Enter a new line of text, or press ENTER.)*

Press ENTER to keep each option as is. If you select more than one line of text, AutoCAD repeats the prompts for the next one.

Attribute Definitions

The **Change Point** option changes the text and properties of attributes that are not part of a block.

- Specify new text insertion point: *(Pick a new insertion point, or press ENTER.)*
- Enter new text style <current>: *(Enter a different text style, or press ENTER.)*
- Specify new height <current>: *(Enter a new height, or press ENTER.)*
- Specify new rotation angle <current>: *(Enter a new angle, or press ENTER.)*
- Enter new text <current>: *(Enter new text, or press ENTER.)*
- Enter new tag <current>: *(Enter a new tag, or press ENTER.)*
- Enter new prompt <current>: *(Enter a new prompt, or press ENTER.)*
- Enter new default value <current>: *(Enter a new default value, or press ENTER.)*

Press ENTER to keep each option in place.



EXPLODE AND XPLODE

The **EXPLODE** and **XPLODE** commands reduce compound objects to their simplest forms.

The commands “break down” blocks, polylines, and other compound objects into basic lines, arcs, and text. Some objects must be exploded several times before they are finally reduced to vector primitives.

TUTORIAL: EXPLODING COMPOUND OBJECTS

1. To explode one or more compound objects, start the **EXPLODE** command:
 - In the ribbons's 2D Home tab, pick the **Explode** button in the Modify panel.
 - At the 'Command:' prompt, enter the **explode** command:
Command: **explode** *(Press ENTER.)*
 - Alternatively, enter the **x** alias at the 'Command:' prompt.
2. In all cases, AutoCAD prompts you to select the compound objects you want to explode:
 - Select objects: *(Pick one or more objects.)*
 - Select objects: *(Press ENTER to end object selection.)*

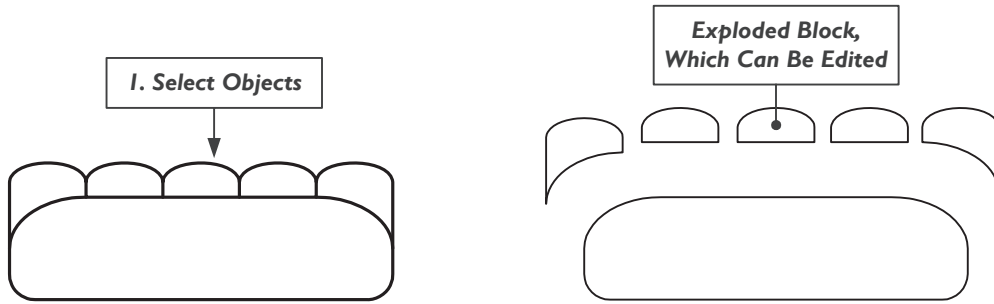
The objects are exploded, but you might not see any difference. Different compound objects react differently to being exploded.

Arcs do not explode, unless they are part of a nonuniformly scaled block. When the block is exploded, arcs become elliptical arcs.

Circles do not explode, unless they are part of a nonuniformly scaled block. When the block is exploded, circles become ellipses.

Blocks explode into the lines and other objects originally used to define them. Nested blocks must be exploded a second time. Attribute text is deleted; attribute definitions are redisplayed.

Xrefs and **blocks** inserted with the **MINSERT** command cannot be exploded.



Left: Block before being exploded.

Right: After exploding, with parts pulled apart for clarity.

Polylines explode into lines and arcs. Width and tangent information is discarded; the resulting lines and arcs follow the center line of the old polyline. If the exploded polyline has segments of width, AutoCAD displays the message:

Exploding this polyline has lost width information.

The UNDO command will restore it.

The new lines and arcs are placed on the same layer as the polyline, and inherit the same color.



Left: Polyline with variable width before being exploded.

Right: Polyline after being exploded, with lines and arcs pulled apart for clarity.

Three-dimensional polylines explode into line segments. Linetypes are retained.

Associative dimensions explode into basic objects (lines, polyline arrowheads, and text), which are placed on the same layer as the original dimensions, and inherit that layer's properties.

Constraints explode into nothingness: when constrained objects are exploded, the constraints are erased.

Leaders explode into lines and multiline text; the arrowheads explode into 2D solids. Depending on how the leader was constructed, resulting objects could also include splines, block inserts of arrowheads and annotation blocks, and tolerance objects. More than one explode may be required.

Multiline text explodes into single-line text.

Multilines explode into lines and arcs.

Single-vertex polyface meshes explode into point objects. Two-vertex meshes explode into lines. Three-vertex meshes, into 3D faces.

Tables explode into lines and mtext.

3D solids explode into bodies, which can be further exploded into regions. Regions, in turn, explode into lines, arcs, and circles.

Legacy mesh objects explode into 3D faces; **3D mesh objects** also explode into 3D faces.

Surfaces explode into lines, arcs, circles, and polylines.



Note When a block is inserted with different scale factors in the x, y, and/or z directions, it is called a “nonuniformly scaled block.” When exploded, the results may be different from what you expect. Arcs, for example, are converted to elliptical arcs to maintain their distortion.

Sometimes, nonuniformly scaled blocks contain objects that cannot be exploded, such as bodies, 3D solids, and regions. In that case, AutoCAD places them in an *anonymous* block with the *E prefix in the block’s name.

XPLODE

The XPLODE command controls what happens when compound objects are exploded.

TUTORIAL: CONTROLLED EXPLOSIONS

1. To explode compound objects with control, start the **XPLODE** command:
 - At the ‘Command:’ prompt, enter the **xplode** command.
Command: **xplode** (Press ENTER.)
2. In all cases, AutoCAD prompts you to select the compound objects you want to explode:

Select objects to Xplode.

Select objects: (Pick one or more objects.)

Select objects: (Press ENTER to end object selection.)
3. Enter an option:

Enter an option

[All/Color/Layer/LType/LWeight/Inherit from parent block/Explode] <Explode>: (Enter an option.)
4. If you had selected more than one object, an additional prompt appears:

Enter an option [Individually/Globally] <Globally>: (Enter I or G.)

The **Global** option applies the previous options (color, layer, and so on) to all selected objects. The **Individually** option applies the previous options to each object, one at a time.

All

The **All** option prompts you to specify the color, linetype, lineweight, and layer on which the exploded objects should land. The prompts are the same as those for the four following options:

Color

The **Color** option sets the color of exploded objects:

[Red/Yellow/Green/Cyan/Blue/Magenta/White/BYLayer/BYBlock/Truecolor/Colorbook]
<BYLAYER>: (Enter a color name or number.)

Layer

The **Layer** option sets the layer of the exploded objects:

Enter new layer name for exploded objects <current>: (Enter the name of an existing layer.)

LType

The **LType** option sets the linetype of the exploded objects:

Enter new linetype name for exploded objects <BYLAYER>: (Enter the name of a linetype.)

Inherit from Parent Block

The **Inherit from parent block** option sets the color, linetype, lineweight, and layer of the exploded objects to that of the originating block.

Explode

The **Explode** option reduces compound objects in the same manner as the **EXPLODE** command.

PEDIT

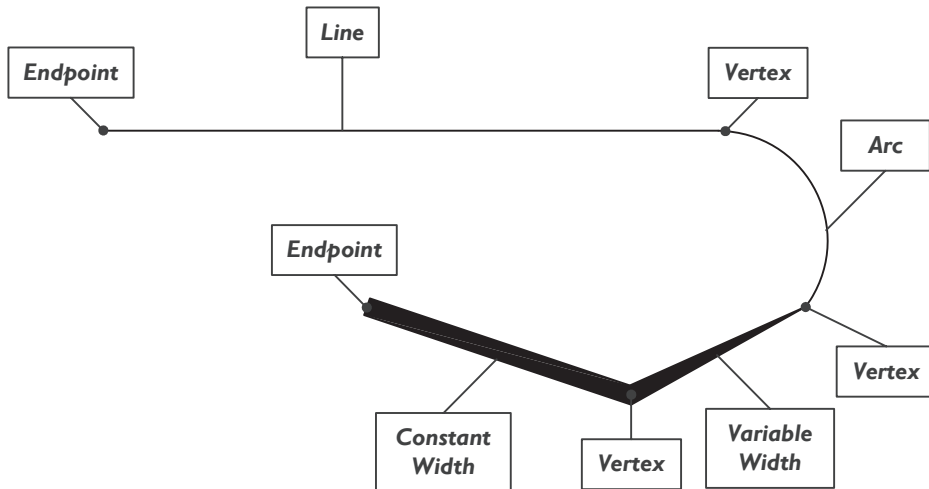
The **PEDIT** command (short for “polyline edit”) edits polylines.

This command also edits 3D polylines drawn with the **3DPOLY** command, and 3D polyfaces drawn with AutoCAD’s 3D surfacing commands.



AutoCAD 2010 adds the **Reverse** option, which operates in the same way as the **REVERSE** command described in the previous chapter: it reverses the direction of polylines. The meaning of the **Spline** command is changed: as of AutoCAD 2010, it converts polylines into true spline objects. (Previously, it smoothed the polyline.) Also new in AutoCAD 2010 is the ability to edit segments of polylines directly.

As a reminder, here are the parts of a polyline:



TUTORIAL: EDITING POLYLINES

- To edit one or more polylines, start the **PEDIT** command:
 - In the ribbon’s 2D Home tab, pick the **Edit Polyline** button in the Modify panel.
 - At the ‘Command:’ prompt, enter the **pedit** command:

Command: **pedit** (Press ENTER.)
 - Alternatively, enter the **pe** alias at the ‘Command:’ prompt.
- In all cases, AutoCAD prompts you to select the polyline you want edited:

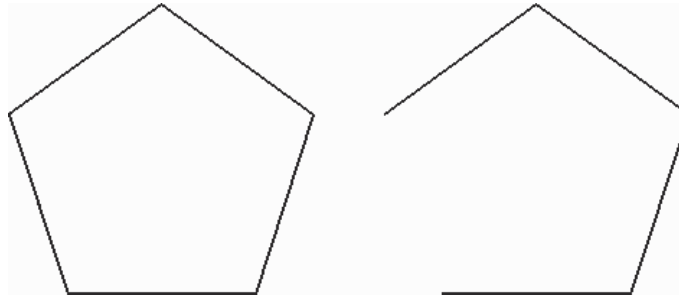
Select polyline or [Multiple]: (Pick a polyline.)
- Many options are listed:

Enter an option [Close/Join/Width/Edit vertex/Fit/**Spline**/Decurve/Ltype gen/**Reverse**/Undo]: (Enter an option, or press esc to exit the command.)

Close/Open

The **Close** option closes open polylines. It connects the last point to the first point of the polyline. This is like the **PLINE** command's **Close** option, except it allows you to close the polyline after exiting **PLINE**.

The **Open** option replaces the **Close** option when the polyline is closed. It opens closed polylines by removing the last segment created by the **Close** option. If the polyline looks closed, but was not closed with **Close**, the **Open** option has no effect.



Left: Closed polyline.

Right: Open polyline.

Join

The **Join** option converts and connects polylines and non-polyline objects (lines and arcs only) to the original polyline. AutoCAD prompts:

Select objects: *(Select one or more lines, arcs, and polylines.)*

The objects you select become part of the original polyline; if the polyline was curve-fitted, it is first decurved.

AutoCAD determines which arcs and lines share common endpoints with the original polyline, and then merges them into that polyline. To join successfully, the objects must touch, or be within a *fuzz factor* distance, as described later in this chapter. If endpoints are too far away to join, use the **FILLET** or **CHANGE** command to extend them to a perfect match. For alternatives, see the **Multiple** and **Jointype** options discussed later in this chapter.

Width

The **Width** option changes the width of the entire polyline. Unlike the **PLINE** command's **Width** option, you can't have tapers or different widths for each segment; the width is applied uniformly to all segments of the polyline. AutoCAD prompts:

Specify new width for all segments: *(Enter a value.)*

You can type the width at the keyboard, or else pick two points in the drawing. The figure illustrates the effect of this option:

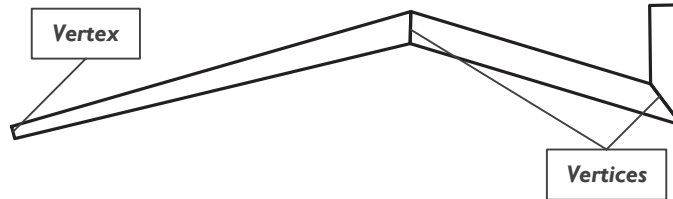


Left: A polyline with varying widths.

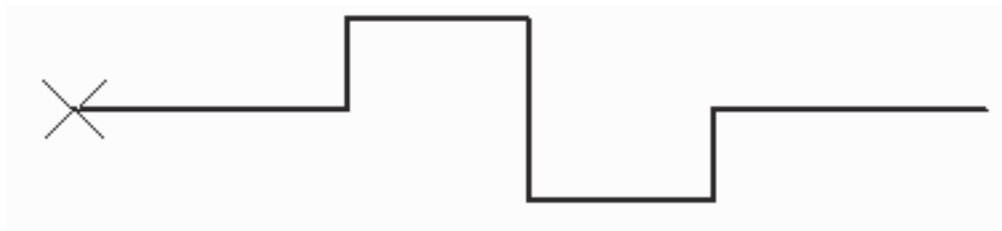
Right: The polyline after applying the **Width** option.

Edit vertex

The suboptions of the **Edit vertex** option edit the vertices of a polyline. (Recall that the vertex is the connection between polyline segments, as well as between the two endpoints of open polylines.) The name, “edit vertex,” is somewhat misleading, because it also edits the segments between pairs of vertices.



To keep track of which vertex is being edited, AutoCAD displays a marker in the shape of an X. Press **N** and **P** to move the marker to the next and previous vertex.



The **x** marks the vertex. Pressing **N** and **P** moves the marker from vertex to vertex.

A second marker, an arrow, is displayed when you work with the **Tangent** option.

Command: **pedit**

Select polyline or [Multiple]: *(Pick a polyline.)*

Enter an option [Close/Join/Width/Edit vertex/Fit/Spline/Decurve/Ltype gen/Undo]: **e**

[Next/Previous/Break/Insert/Move/Regen/Straighten/Tangent/Width/eXit]: *(Enter an option.)*



Note Quite frankly, the **Edit vertex** options are a pain to use. In many cases, it much easier to edit the polyline using grips. Use these options only to insert and delete vertices. Use the Properties palette to set the widths of individual segments.

Width

The **Width** option changes the starting and ending widths for the segment immediately following the marked vertex. AutoCAD prompts:

Specify starting width for next segment: *(Enter a width, or pick two points.)*

Specify ending width for next segment: *(Enter a width, or pick two points.)*

Use the **Regen** option to see the new width.

Exit

The **Exit** option exits **Edit vertex** mode, and returns to **PEDIT**'s original prompt line.

THE MANY SUB-OPTIONS OF EDIT VERTEX

Next moves the x-marker to the next vertex, while **Previous** moves it to the previous vertex.

Break removes one or more segments between two vertices. AutoCAD places the x-mark at the first vertex on one side of the break. You mark the other vertex, on the other side of the break, using the following options:

Enter an option [Next/Previous/Go/eXit]: *(Enter an option.)*

Press **N** and **P** to move the x-marker to another vertex, and then press **G** to effect the break. Press **X** to exit this mode without breaking the polyline. The entire segment is removed; to remove a portion of a segment, use the **BREAK** command. This option does not work under two conditions: when the same vertex is marked twice, so that no segment is selected for removal; or when the polyline's two endpoints are marked, in effect removing the entire polyline.

Insert adds a vertex to the polyline at your pick point. AutoCAD prompts:

Specify location for new vertex: *(Pick a point anywhere in the drawing.)*

AutoCAD redraws the polyline so that it includes the new vertex.

Move moves the marked vertex. AutoCAD prompts:

Specify new location for marked vertex: *(Pick a point anywhere in the drawing.)*

Regen regenerates the polyline, without requiring you to exit the command. This option is required after applying the **Width** option.

Straighten removes vertices, line segments, and arcs between two marked vertices, and replaces them with a single segment. It works like **Break**: AutoCAD remembers the x-marked vertex before you enter this option, and then you move the x-marker to another vertex.

Tangent attaches a tangent direction to the marked vertex for use later with the **Fit** and **Spline** options. AutoCAD prompts:

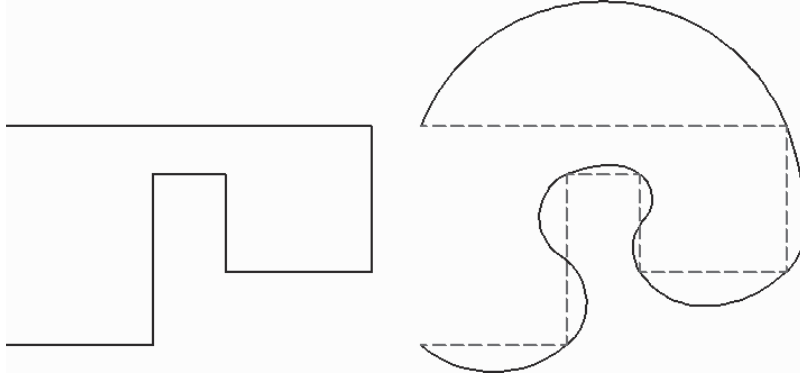
Specify direction of vertex tangent: *(Pick a point, or enter an angle.)*

AutoCAD displays an arrow pointing in the direction of tangency.



Fit

The **Fit** option constructs smooth curves from the vertices in the polyline. The curve consists of arcs joined at vertices, as illustrated by the figure. Notice how each segment has at least one arc associated with it. AutoCAD inserts extra vertices and arcs, where necessary; in the figure, two segments have two arcs each.



Left: Original polyline.

Right: Polyline after applying the **Fit** curve option; the original polyline shown overlaid by gray dashed lines.



Spline

The **Spline** option converts the polyline into a true spline object. (Prior to AutoCAD 2010, the option turned polylines into an approximated spline, using the vertices as the control points to approximate a B-spline curve; if the original polyline contained arcs, they were converted to straight segments.) The **SPLINEDIT** command reverses the process, converting splines into polylines.

Decurve

The **Decurve** option negates the effect of the **Fit curve** option by straightening all segments of the polyline, as well as by removing extra vertices that may have been added.

Ltype gen

The **Ltype gen** option (short for “Linetype generation”) controls the generation of linetypes through the vertices of the polyline. Normally, AutoCAD generates the dashes, dots, and gaps for each segment of the polyline; the pattern starts and stops at each vertex.

Enter polyline linetype generation option [ON/OFF] <Off>: (Enter **ON** or **OFF**.)

When the option is turned on, the polyline is treated as a single segment, resulting in a uniform linetype pattern along the entire length of the polyline. In most cases, you probably want this option turned on.



Left: Polyline with **Ltype gen** turned off, part of the polyline lacking the line pattern.

Right: Polyline after turning on the option, all of the polyline patterned..



Note Linetype generation does not work on polylines with tapered segments.



Reverse

The **Reverse** option operates in the same way as the **REVERSE** command described in the previous chapter: it reverses the direction of polylines.

Undo

The **Undo** option reverses the last **PEDIT** operation without requiring you to exit the command to access the **U** command.

EDITING POLYLINES: ADDITIONAL METHODS

Of the **PEDIT** command's many options, some are not obvious:

- **Select** option selects a line or arc.
- **Multiple** option selects more than one polyline.
- **Jointype** option determines how multiple objects are joined into a single polyline.

Let's look at each.

Select

The "Select polyline" prompt is misleading, because you can also select a non-polyline object — specifically a line or an arc. In that case, AutoCAD notices:

Command: **pedit**

Select polyline or [Multiple]: *(Pick a line or arc.)*

Object selected is not a polyline.

Do you want to turn it into one? <Y>: *(Enter Y or N.)*

When you respond with "Y", the object is converted to a polyline. If you had selected an object other than a line or arc, the conversion fails, and AutoCAD complains:

Object selected is not a polyline.

Multiple

You can select more than one polyline to edit, but only by entering the **Multiple** option, as follows:

Command: **pedit**

Select polyline or [Multiple]: **m**

Select objects: *(Pick one or more polylines.)*

Select objects: *(Press ENTER to end polyline selection.)*

Jointype

The **Join** option merges lines, arcs, and polylines into a single polyline, but they all need to be touching. When you use the **Multiple** option at the start of the **PEDIT** command, however, then the objects *need not touch*! AutoCAD instead displays the following prompt:

Join Type = Extend

Enter fuzz distance or [Jointype]<0.0000>: *(Enter a distance, or type J.)*

The *fuzz distance* determines how far apart the objects can be, and still be joined to the original polyline. The **Jointype** option specifies how distant objects should join:

Enter a vertex editing option

Enter join type [Extend/Add/Both] <Extend>: *(Enter an option.)*

The **Extend** option joins the selected objects by extending and trimming them to fit.

The **Add** option joins the selected objects by bridging them with a straight segment.

The **Both** option joins the selected objects by extending or trimming; if that is not possible, AutoCAD adds the straight segment.

When you attempt to join two non-touching polylines, AutoCAD complains:

0 segments added to polyline

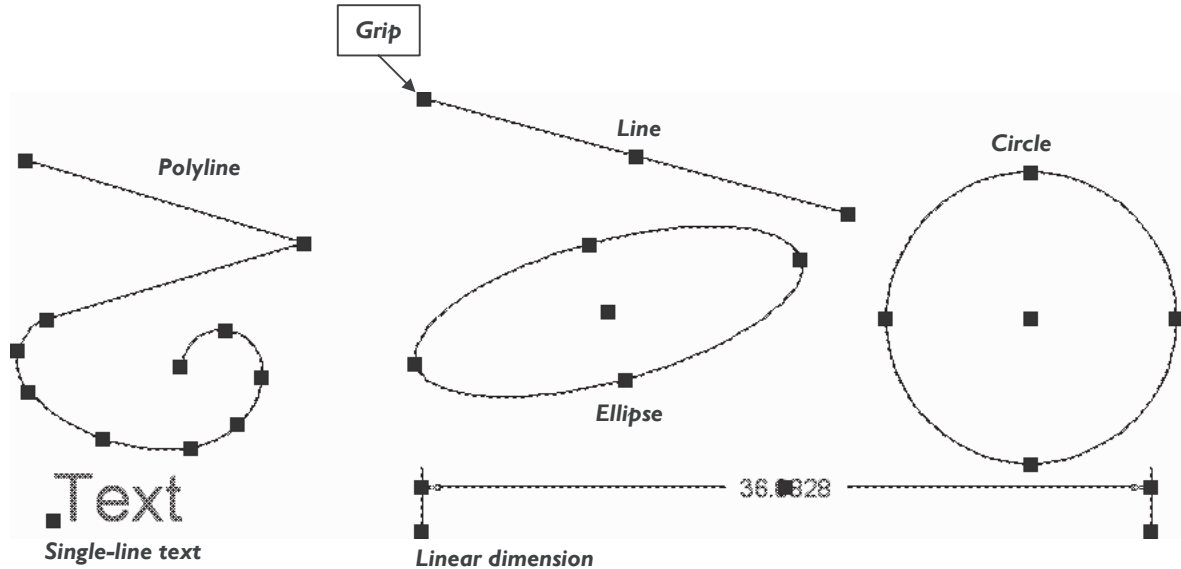
No matter which option you pick, fit-curved and splined polylines lose their curvature, and are converted back to the original frame.

GRIPS

Grips allow direct editing of objects — without commands.

In traditional AutoCAD usage, you enter an editing command and then select the objects to edit. The reverse is also possible: first select the objects, and then edit them directly without entering commands.

Grips are small, colored squares that indicate where the object can be edited. For example, lines have a grips at either end, plus another at their midpoint; circles have grips at the center point and at the four quadrant points.



Grips editing can be quick. For example, a quick way to erase objects is to select them with the cursor, and then hit the DEL key.



Note To select all objects in the drawing, press **CTRL+A** (short for “all”) at the command prompt:

Command: (Press **CTRL+A**.)

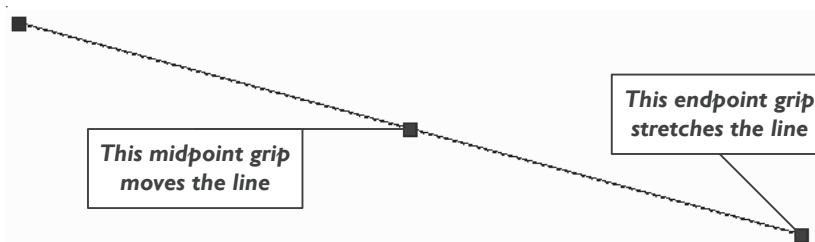
AutoCAD selects everything in the drawing, except those on frozen and locked layers.

DIRECT EDITING WITH GRIPS

Once an object is selected, you can perform two editing operations with it, in most cases:

Move — drag the center grip to move the object.

Stretch — drag one of the other grips to make the object, larger or smaller.



To move objects using grips, follow these steps:

- 1. Click on the object to select it. Notice the highlighting and the grips.
- 2. Click on the center grip. Notice that it turns red.
- 3. Move the mouse; don't drag! (To move orthogonally, hold down the **SHIFT** key.)
- 4. Click to place the moved object in its new location.
- 5. When done, press the **ESC** key to unselect the object.

The same actions apply to stretching objects, except that you select one of the other grips.

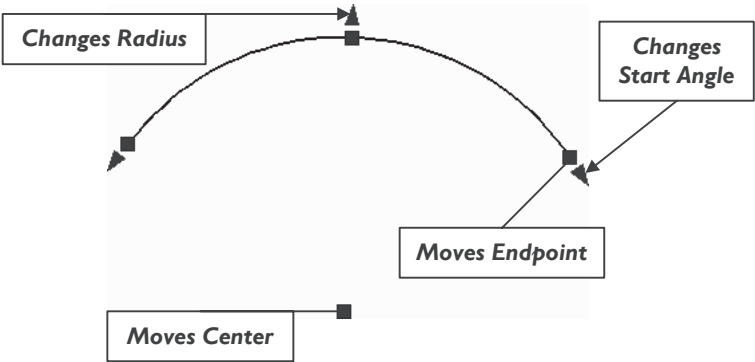
Grip Color and Size

As a visual aid, standard grips are assigned colors and names:

Grip Color	Name	Meaning
Blue	Cold	Grip is not selected.
Cyan	Dynamic	Grip's actions are defined by the Block Editor.
Pink	Hover	Cursor is positioned over grip.
Red	Hot	Grip is selected, and object can be edited.

Grips on Arcs

Arcs have additional grips not found on other objects. These extra grips are triangular in shape and perform specific editing tasks, as illustrated below.



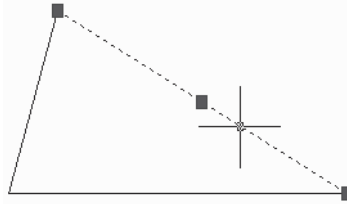
- Square grips move the arc and its endpoints.
- Triangular grips change the angle of the start and endpoints, and the radius of the arc.

TUTORIAL: EDITING WITH GRIPS

Although you can move and stretch objects directly through grips, AutoCAD provides a number of additional commands that you select at the command prompt. These are Stretch, Move, Rotate, Scale, Copy, and Mirror.

After objects are selected, these appear as options on the command line and the shortcut menu. In addition to these special six, you can use other AutoCAD commands with grips. Examples include ERASE, COPYCLIP, PROPERTIES, MOVE, SCALE and STRETCH. Some editing commands, such as TRIM and OFFSET, cannot be used with grips.

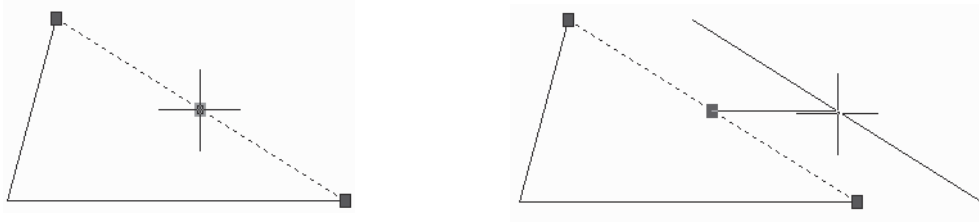
1. Select an object in the drawing. The object is highlighted.



Notice that one or more blue grips are placed on the object (sometimes called “cold”). You can select as many objects as you wish by selecting more than once. If you cannot select more than one object, hold down the **SHIFT** key on the keyboard when selecting the grips.

2. Move the cursor over a blue grip.

As the cursor gets close, notice that it jumps to the grip, and that the grip changes color to pink. The grip is said to be “warm” and to “hover.”



Left: Clicking a grip makes it hot.

Right: Dragging the grip moves or stretches the object.

3. Click the pink grip.

Notice that the grip box changes color to red to denote selection (called “hot”). The hot grip is the base point from which some editing actions, such as stretch and rotate, take place.

4. Notice that at the command line, AutoCAD prompts:

**** STRETCH ****

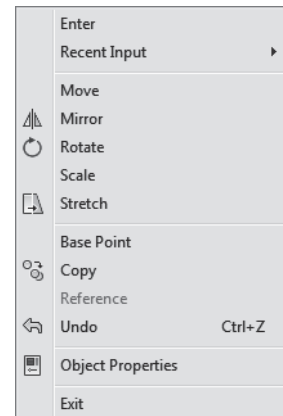
Specify stretch point or [Base point/Copy/Undo/eXit]:

You can now drag the grip to stretch or move the object, depending on which grip is red.

Or, press the spacebar to see the other editing options: Move, Rotate, Scale, and Mirror. These are described next.

You can right-click a hot grip to see these editing options.

5. Press **ESC** once to clear the hot grip and selected objects.



GRIP EDITING MODES

When you select a single grip (so that it turns red), AutoCAD displays editing options on the command line. The options cycle as you press the spacebar or **ENTER**. (Right-click to see the shortcut menu listing the same options.)

**** STRETCH ****

Specify stretch point or [Base point/Copy/Undo/ eXit]: (*Press spacebar.*)

**** MOVE ****

Specify move point or [Base point/Copy/Undo/ eXit]: *(Press spacebar.)*

**** ROTATE ****

Specify rotation angle or [Base point/Copy/Undo/Reference/eXit]: *(Press spacebar.)*

**** SCALE ****

Specify scale factor or [Base point/Copy/Undo/Reference/eXit]: *(Press spacebar.)*

**** MIRROR ****

Specify second point or [Base point/Copy/Undo/ eXit]: *(Press spacebar.)*

Press the spacebar until the desired edit option is listed, and then proceed with the command. For example, to resize the object, press the spacebar until ****SCALE**** appears, and then enter a scale factor at the “Specify scale factor:” prompt.

The selected grip point becomes the base point for editing. Alternatively, select one of the options listed on the command line. The commands and their options are covered later in this section.



Note It is possible to work with two hot grips at a time, notes the technical editor. For instance, draw two lines, and then select both. While holding down the **SHIFT** key, select two (or more) grips. Release the **SHIFT** key, and then drag one of the hot grips. As you do, the other line also moves along.

GRIP EDITING OPTIONS

Let's look at each grip editing option.

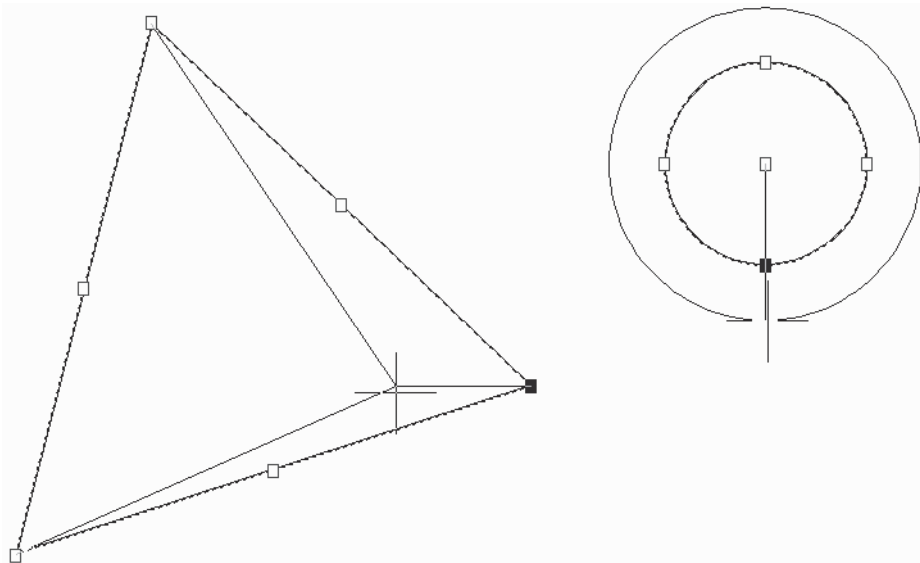
Stretch Mode

Stretch functions like the **STRETCH** command, except that AutoCAD uses the hot grip to determine the stretch results.

**** STRETCH ****

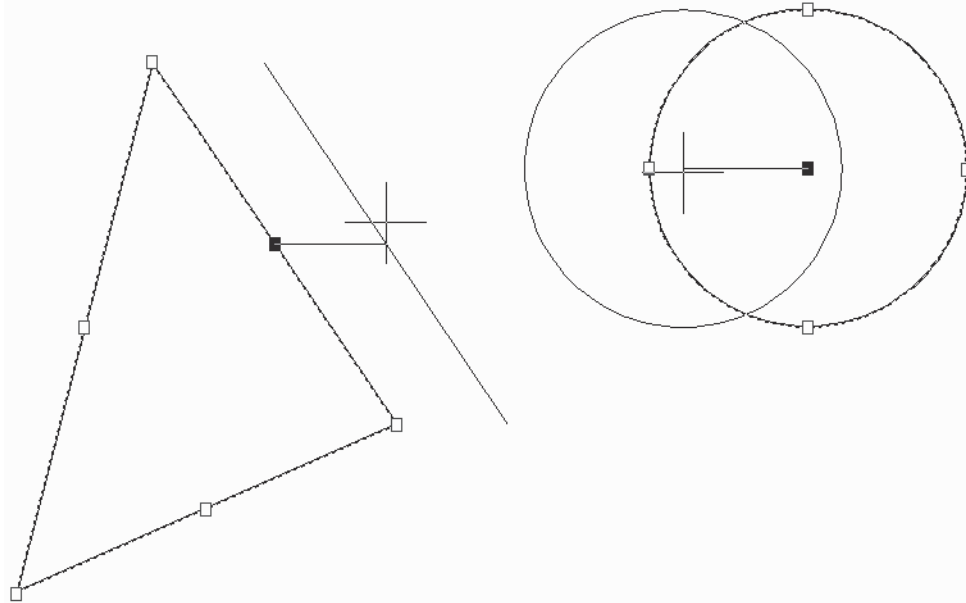
Specify stretch point or [Base point/Copy/Undo/eXit]: *(Pick a point, or enter an option.)*

Stretch. When you select the end grip of a line or arc, or quad grip of a circle, the object is *stretched*. Lines and arcs “stretch” longer or shorter, while circles change their diameter. The figure illustrates stretching these objects. (For clarity, the hot grips are shown as black squares, and the cold grips as white.)



The triangle is made of lines, so AutoCAD is stretching two lines at once, because of their common endpoints. The circle is not being stretched into an ellipse, as you might think from the word “stretch;” instead, its diameter is being changed.

Move. When you select the midpoint grip of a line, or arc, or the center of a circle, the object is *moved*, but not stretched. In the figure, one line of the triangle is being moved, because the grip is at the center of the line; the entire circle is being moved.



Move Mode

Move moves the selected objects. All selected objects are moved, relative to the hot grip.

**** MOVE ****

Specify move point or [Base point/Copy/Undo/eXit]: *(Pick a point, or enter an option.)*

Objects move by the distance from the base point to the point you pick. You can either pick a point, or enter absolute or relative coordinates to specify the distance and direction of the move. For example, enter relative coordinates that specify a distance of 50 units at an angle of 45 degrees from the x-axis:

Specify move point or [Base point/Copy/Undo/eXit]: **@50<45**

Rotate Mode

The **Rotate** option rotates the selected objects around the base point. All selected objects are rotated, relative to the hot grip.

**** ROTATE ****

Specify rotation angle or [Base point/Copy/Undo/ Reference/eXit]: *(Pick a point, or enter an option.)*

Rotation occurs around the hot grip, unless you use the **Base point** option to relocate it. When rotating objects, you can dynamically set the rotation angle by moving the crosshairs, or by specifying a rotation in degrees.

Scale Mode

If you wish to resize the selected objects, cycle through the mode list until **Scale** is shown on the command line. All selected objects are resized, relative to the hot grip.

**** SCALE ****

Specify scale factor or [Base point/Copy/Undo/ Reference/eXit]: *(Pick a point, or enter an option.)*

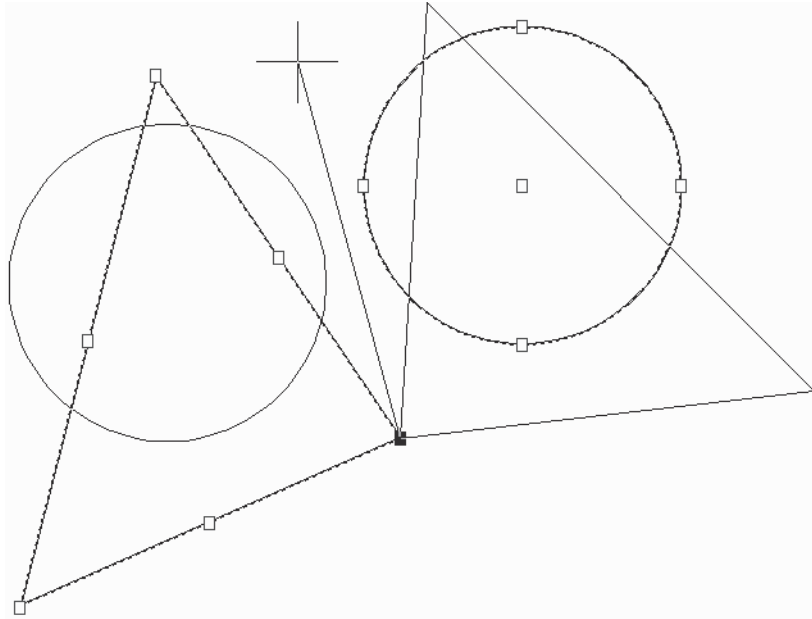
The hot grip serves as the base point for resizing. Dynamically resize the objects by moving the crosshairs away from the base grip. You can also resize the selected objects by entering a scale factor.

Mirror Mode

Mirror mode mirrors the selected object(s) relative to the hot grip. This option differs from the **MIRROR** command in that the objects are moved — not copied! — about the *mirror* line. The mirror line is created by the cursor, with the first point being established by the hot grip (unless you change it with the **Base point** option). AutoCAD prompts you for the second point of the mirror line:

**** MIRROR ****

Specify second point or [Base point/Copy/Undo/ eXit]: *(Pick a point, or enter an option.)*



Grip Editing Options

Each grip editing option has a set of common suboptions.

Base point

Normally, the hot grip is the base point for the editing operation. To change it, enter **B**, and AutoCAD prompts you:

Specify base point: *(Pick a point.)*

The point you pick becomes the new base point.

Copy

Enter **C** to copy the selected object, leaving the original intact. The prompt changes to:

**** STRETCH (multiple) ****

Specify stretch point or [Base point/Copy/Undo/eXit]: *(Pick a point.)*

The selected object is copied to the point you pick. The prompt repeats itself so that you can make multiple copies, as indicated by the word “(multiple).” Press **ENTER** or **ESC** to exit the command.

Undo

Enter **U** to undo the last operation.

Reference

Enter **R** to provide a reference for the Scale and Rotate modes. For Scale mode, AutoCAD prompts:

Specify reference length <1.0000>: *(Enter a length, or pick two points.)*

Sometimes the Reference option can be difficult to understand. Here is an example. Suppose you have an object that is 6 units in length, but you wish to resize it to 24 units. Select the **Reference** option, and then enter original length, as follows:

Specify scale factor or [Base point/Copy/Undo/Reference/eXit]: **r**

Reference length <1.0000>: **6**

AutoCAD then prompts for the new length.

<New length>/Base point/Copy/Undo/Reference/eXit: **24**

AutoCAD calculates the scale factor, and then resizes the object.

For **Rotate** mode, AutoCAD prompts:

Specify reference angle <0>: *(Enter an angle, or pick two points.)*

The reference angle is the angle at which the object is currently rotated. AutoCAD next prompts for the “New angle.” The new angle is the angle you want to rotate the object.

eXit

Enter **X** or press **ESC** to exit grips editing.

Other Grip Editing Commands

You can also use other commands when objects are highlighted with grips. Here is a sample:

ERASE deletes the gripped objects from the drawing. As an alternative, press the **DEL** key. *Caution:* selecting all objects and then pressing **DEL** erases the entire drawing! Use **U** to recover.

COPYCLIP copies the gripped objects to the Clipboard; **CUTCLIP** cuts the objects. As alternatives, you can press **CTRL+C** and **CTRL+X**, respectively.

PROPERTIES displays the Properties palette for the gripped objects.

MATCHPROP copies the properties of one gripped object; if more than one object is gripped, AutoCAD complains, “Only one entity can be selected as source object.”

ARRAY creates rectangular and polar arrays of the gripped objects.

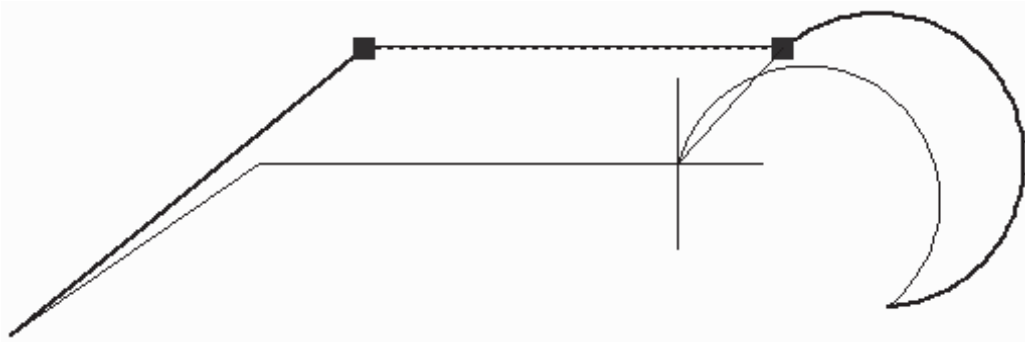
BLOCK creates a block of gripped objects.

EXPLODE explodes gripped objects; if they cannot be exploded, AutoCAD complains, “1 was not able to be exploded.”

Grips Editing Polyline Segments

You can edit polylines with grips as you can lines, except that polyline segment has just one grip at either end. You can also edit an individual polyline segment with grips ([new to AutoCAD 2010](#)):

1. Hold down the **CTRL** key.
2. Select a segment of the polyline. Notice that only the segment is highlighted, and that it has two red grips at either end.
3. Select one of the grips.



4. Move the grip, or press the **SPACEBAR** to cycle through the grip editing modes described later: Stretch, Move, Scale, Rotate, or Mirror. (Copy is not available.)
5. Press **ESC** to exit grips editing mode.

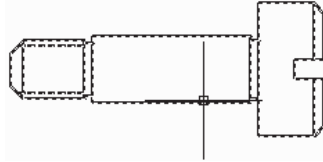
Alternatively, you can enter the **MOVE**, **ROTATE**, or **SCALE** command, and then hold down the **CTRL** key as you select the segment to edit.

RIGHT-CLICK MOVE/COPY/INSERT

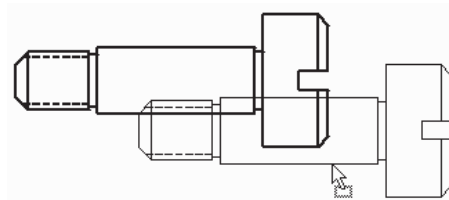
A simpler alternative to grips editing is *right-click move/copy/insert*. It is meant as a faster method to copy and move objects, as well as to insert them as blocks. (This action is not documented by Autodesk, so I don't know its official name.)

It works like this:

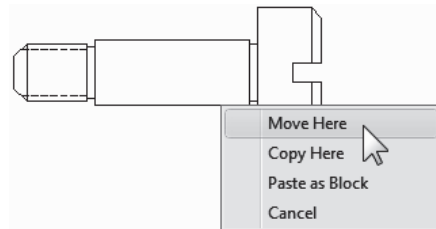
1. Select one or more objects.



2. Holding down the right mouse button, drag the object. (*Important!* Drag the object, not a grip.) Notice the special cursor icon (small rectangle).



3. Let go of the mouse button. AutoCAD displays a small shortcut menu with these options:



Option	Meaning
Move Here	Moves the selected objects.
Copy Here	Copies the selected objects, and locates them.
Paste As Block	Turns the objects into a block, and then inserts them.
Cancel	Cancels the operation; the objects remain as they were.

4. Choose an option from the shortcut menu.

When you choose the Paste As Block option, AutoCAD names the block automatically with a name like "A\$C39FB63B0". Use the **RENAME** command to change the block's name to something more meaningful.

DIRECT DISTANCE ENTRY

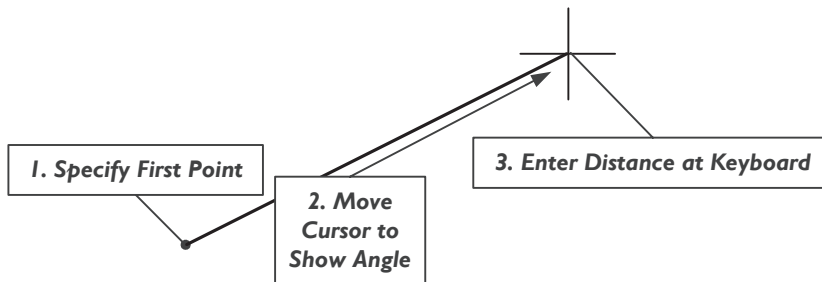
Direct distance entry allows “pen up” movement during drawing and editing commands. (*Pen up* means that the cursor changes its position without drawing or editing objects.)

Direct distance entry is an alternative to entering polar or relative coordinates that rely on a distance and an angle. To show the angle, you move the mouse; then you type the distance. With ortho or polar modes turned on, direct distance entry is an efficient way to draw lines.

Direct distance entry can be used any time a command prompts you to specify a point, such as “Specify next point:”.

TUTORIAL: DIRECT DISTANCE ENTRY

1. To draw a line 10 units long using direct distance entry, enter a drawing or editing command:
Command: **line**
2. Before you can use direct distance entry, you must pick an initial point:
Specify first point: (*Pick a point.*)
3. At the next prompt, move the mouse to indicate the angle, and then enter the distance at the keyboard:
Specify next point: (*Move the mouse, and you see a rubber band line. At the keyboard, type 10 and press ENTER.*)



4. End the command:
Specify next point: (Press **ENTER**)

AutoCAD draws a line segment ten units long in the direction you move the mouse. You can use direct distance entry for drawing polylines, arcs, multilines, and most other objects. Direct distance entry does not make sense with some drawing commands, like **CIRCLE** and **DONUT**, because they want a radius.

Additionally, you can use direct distance entry with editing commands, such as **MOVE**, **STRETCH**, and **COPY**.



TRACKING

The **Tracking** modifier allows “pen up” movement during drawing and editing commands.

Whereas direct distance entry lets you draw relative distances, tracking lets you move relative distances within a command. Tracking is not a command, but a command option; you can use tracking only within another command.

Upon entering tracking mode, AutoCAD automatically switches to ortho mode, and then prompts for the “First tracking point:”. (If polar mode is on, AutoCAD switches it off and turns on ortho mode.) Once you exit tracking mode, AutoCAD changes ortho and polar back to their original states.

Some commands keep prompting you for additional points, such as the **LINE** and **PLINE** commands. With these, you can go in and out of tracking mode as often as you like.

TUTORIAL: TRACKING

1. To employ tracking, enter a drawing or editing command:
Command: **line**
2. At a “Specify first point:” or “Specify next point:” prompt, enter **tracking**, **track**, or **tk**.
Specify first point: **tk**
3. AutoCAD enters tracking mode. Notice that ortho mode is turned on. (Look at the **ORTHO** button on the status bar.)
Move the mouse a distance, and then click.
First tracking point: *(Move the mouse and click.)*
4. This time, move the mouse, and then enter a distance.
Next point (Press Enter to end tracking): *(Move the mouse in another direction, and then enter a distance, such as 2.)*
5. Press **ENTER** to exit tracking mode, and resume drawing with the **LINE** command.
If ortho mode was off when you started the line command, AutoCAD switches it off automatically upon exiting tracking mode.
Next point (Press Enter to end tracking): *(Press ENTER.)*
To point: **2,3**
To point: **10,5**
6. Switch back to tracking mode, and enter distances as absolute x, y coordinates:
To point: **tk**
First tracking point: **5,10**
7. Exit tracking mode, draw another line, and exit the **LINE** command.
Next point (Press Enter to end tracking): *(Press ENTER.)*
To point: **3,2**
To point: *(Press ENTER.)*

If you want tracking to move in angles other than 90 degrees, use the **Rotate** option of the **SNAP** command to change the angle. Since the **SNAP** command is transparent, you can change the tracking angle in the middle of tracking, as follows:

Command: **line**

Specify first point: *(Pick.)*

Specify next point: **tk**

First tracking point: *(Move cursor and pick a point.)*

Next point (Press Enter to end tracking): **'snap**

>>Specify snap spacing or [ON/OFF/Aspect/Rotate/Style/Type] <0.5000>: **r**

>>Specify base point <0.0000,0.0000>: *(Press ENTER to accept default.)*

>>Specify rotation angle <0>: **45**

Resuming LINE command.

Next point (Press Enter to end tracking): *(Move cursor and pick a point.)*

Next point (Press Enter to end tracking): *(Press ENTER to end tracking.)*

Specify next point: *(Pick a point.)*

Specify next point: *(Press ENTER to exit command)*



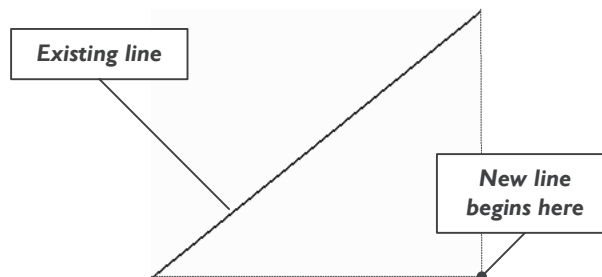
Note Tracking normally assumes you want to switch direction each time you use it. For example, if you first move north (or south), AutoCAD assumes you next want to move east (or west).

It is not easy, however, to back up or track forward in the same direction. For example, if you track north and then want to track north some more, you find the cursor wanting to move east or west, but not north or south. To make the tracking cursor continue in the same direction, take it back to its most recent starting point. Then move in the direction you want.

OSNAP WITH OBJECT TRACKING

Technical editor Bill Fane points out that it is possible to employ tracking without using the **TK** option. AutoCAD can acquire coordinates from other objects through object tracking. Follow these steps:

1. Turn on object tracking: click the **OTRACK** button on the status bar.
2. Turn on **ENDpoint** object snap: right-click the **OSNAP** button on the status bar, and then choose **Settings**. In the dialog box, ensure endpoint is on, and then exit the dialog box.
3. Draw a line at an angle, similar to the one illustrated below.

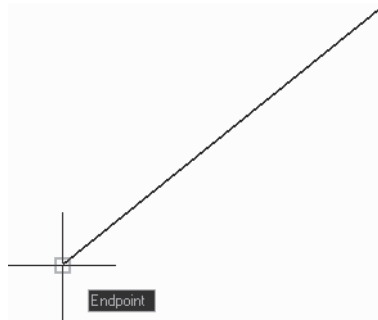


4. You want to draw a second line that starts at the point illustrated above. Enter the **LINE** command:

Command: **line**

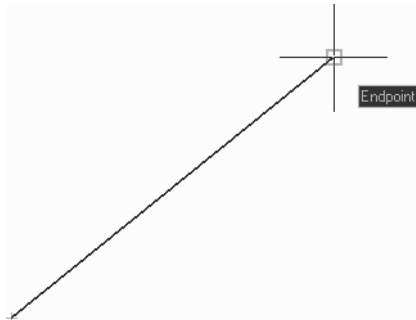
5. At the prompt, move the cursor over the lower endpoint of the existing line. Wait until the tag “Endpoint” appears

Specify first point: *(Move cursor over endpoint.)*



AutoCAD has acquired the coordinates of the endpoint.

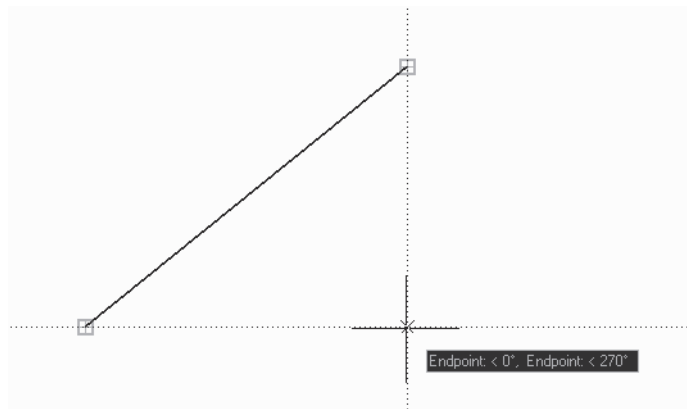
6. Move the cursor to the existing line's other endpoint, and pause. Again, the “Endpoint” tag appears after a moment.



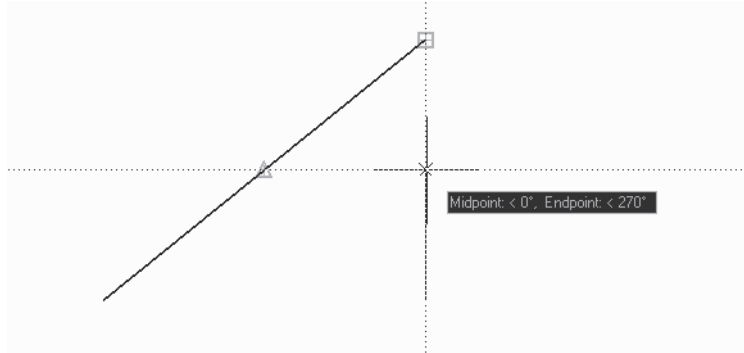
AutoCAD has acquired the coordinates of the other endpoint.

Notice the barely perceptible + marker at the first endpoint. It indicates that AutoCAD is still remembering its coordinates.

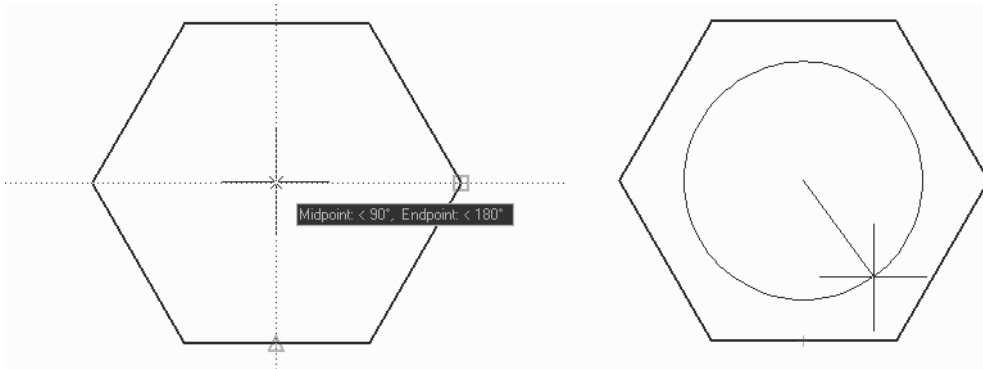
7. Move the cursor to where the new line is to begin, at the “intersection” of the two endpoints. You'll know you've arrived when the object tracking lines appear, as illustrated below.



8. Now click to fix the first point:
Specify first point: *(Pick the point.)*
9. You can continue using this technique to find other points through object tracking, such as the “intersection” of the midpoint and endpoint, as illustrated below.



This technique is not limited to lines, but can be used with any objects, such as placing a circle in the center of a hexagon.

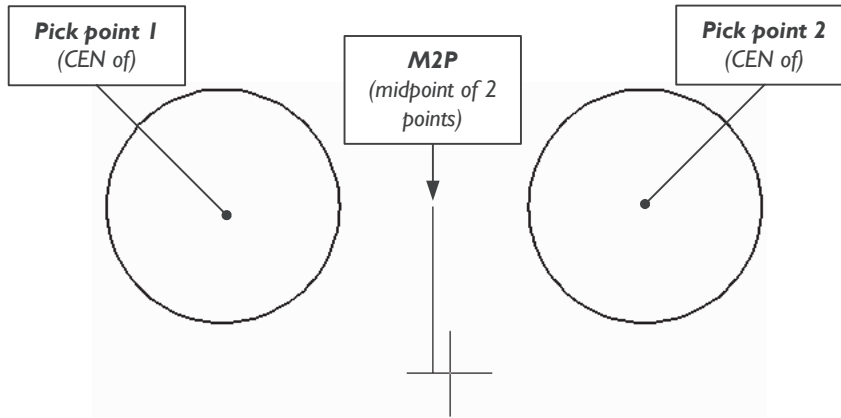


This method also works with editing commands, such as **MOVE** and **COPY**. Object tracking pretty much eliminates the need for construction lines when editing orthographic views.

M2P

The **M2P** modifier finds the midpoint between two picked points — hence “m2p.” (Alternatively, you can enter “mtp.”)

M2P sounds like an object snap, but is not. (Nor it is an early version of music file.) It is entered during any prompt that asks you to specify a point. You can employ object snaps during the M2P process.

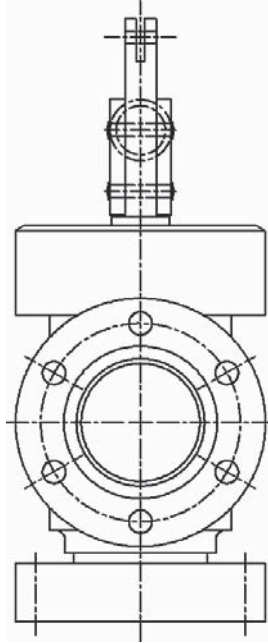


TUTORIAL: M2P

1. To benefit from **M2P**, enter a drawing or editing command:
Command: **line**
2. At a “Specify first point:” or “Specify next point:” prompt, enter **m2p** or **mtp**.
Specify first point: **m2p**
3. AutoCAD prompts you for the first “mid” point. I recommend using object snaps to help place the point accurately. In this example, the **CEN**ter object snap is specified:
First point of mid: **cen**
of (Pick point 1.)
4. AutoCAD prompts you for the second point:
Second point of mid: **cen**
of (Pick point 2.)
5. The **LINE** command carries on. AutoCAD draws the line starting at the midpoint of the two pick points.
Specify next point: (Continue with the command.)

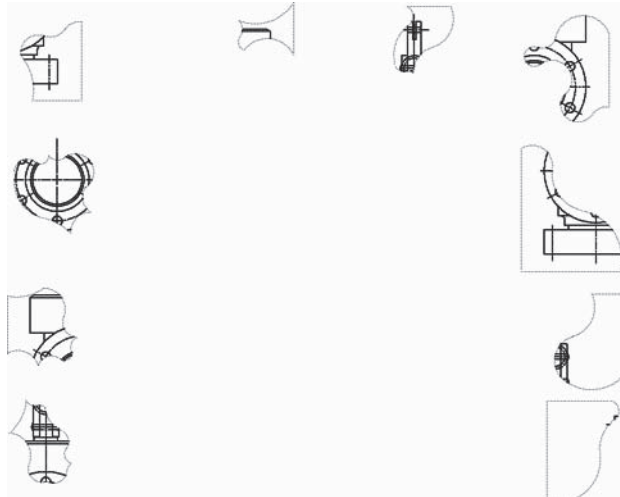
EXERCISES

1. In this exercise, you erase objects from a drawing, and then bring them back.
From the CD, open the *edit1.dwg* drawing file, a valve housing.
Use the **ERASE** command to delete some of the objects.
Next, issue the **OOPS** command. Did the objects return?

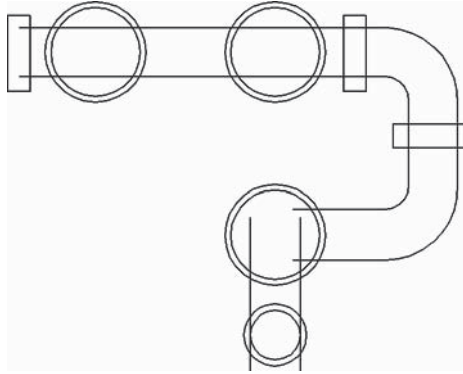


Repeat the **ERASE** command with the **All** option. Did everything disappear?
This time, use the **U** command to bring back the drawing.

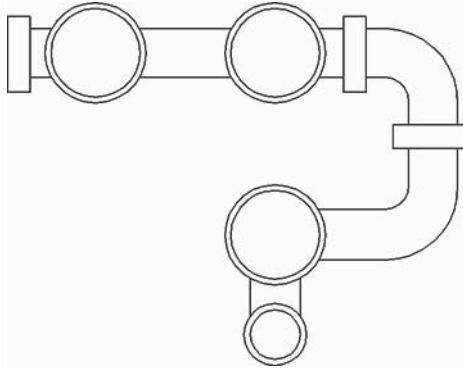
2. In this exercise, you assemble a jigsaw puzzle.
From the CD, open the *edit2.dwg* file, a drawing of puzzle pieces.
Use the **MOVE** command to move the pieces into position, leaving a small space between them. The most effective method is to move the pieces roughly into position, then zoom in, and finely position the pieces using object snap.
You may also want to use grips editing to move pieces into place.



3. In this exercise, you practice breaking and trimming lines.
From the CD, open the *edit5.dwg* drawing file, a piping diagram.



Use the **BREAK** command to break each of the objects in the drawing, achieving the result shown.



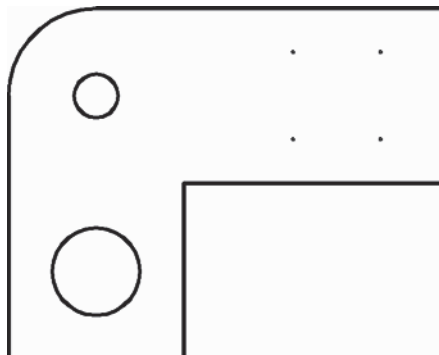
Quit the drawing — don't save your work!

Open *edit5.dwg* again; this time use the **TRIM** command to produce the same result as shown above.

Here's a timesaving tip: at the "Select cutting edges" prompt, just press **ENTER** to select all objects in the drawing.

You may need to use the **ERASE** command to clean up. Which command did you find easier — **BREAK** or **TRIM**?

4. In this exercise, you work with several editing commands.
From the Companion CD, open the *edit7.dwg* drawing file, a piece of sheet metal.



Suppose that you are instructed to remove the lower circle from the drawing.

From the Modify menu, use the **Erase** command.

Now, erase the four points on the object by using a window selection.

Your boss changes his mind.

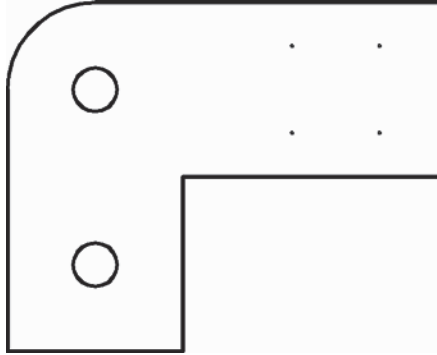
Put back the four points you just erased.

After erasing the lower circle, your boss decides the remaining circle should be moved down by 2.0 units.

Use CENter object snap, ortho mode, and direct distance entry to move it accurately.

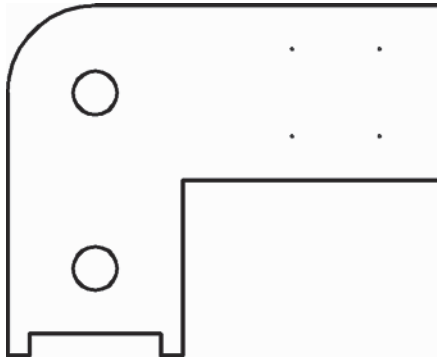
After reviewing the drawing, your boss's boss feels the sheet metal needs two holes.

Add another circle, identical to the remaining circle. Use grips editing to copy the circle.



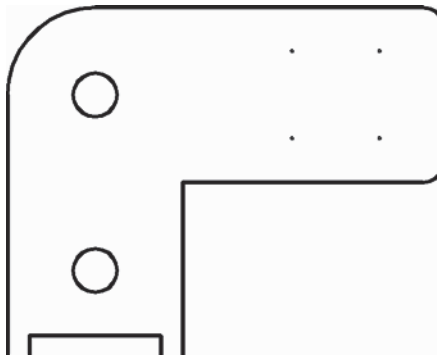
You learn from the product design department that the sheet metal is too large. A notch needs to be taken out.

Turn on snap mode, and set the snap spacing to 0.25. Use the **BREAK** and **LINE** commands to draw the notch.



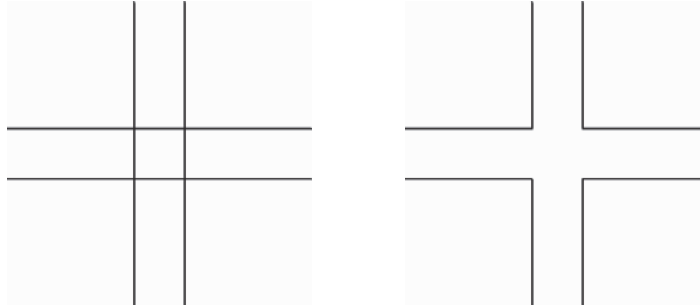
The product safety department reviews the drawing, and feels the sheet metal may be hazardous to children under three years of age. You are asked to round off two corners.

Add a radius of 0.5 to each of the two right corners.



At last, all departments are satisfied. While the scenario in this exercise may seem silly, you will probably experience such changes to your drawings in the workplace.
Save the drawing.

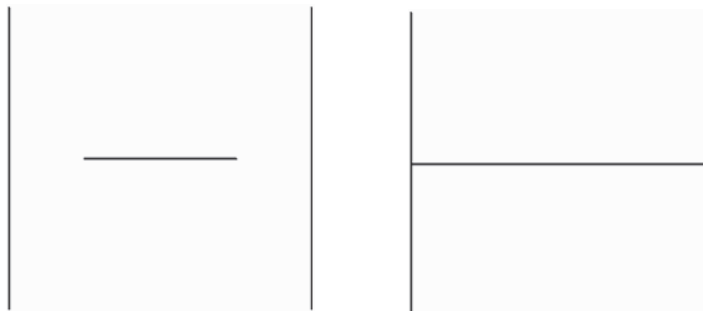
5. In this exercise, you trim objects. Draw four intersecting lines about 6 units long and 2 units apart, as illustrated by the figure. Which command quickly makes the second, parallel line?



Left: Four lines before trimming. **Right:** After trimming.

Use the **TRIM** command to trim the intersections. Remember to use the trick of responding **all** to the “Select cutting edges” prompt.

6. In this exercise, you extend objects. Draw two vertical parallel lines 6 units long and 6 units apart. Between them, draw one horizontal line 4 units long, as illustrated by the figure.

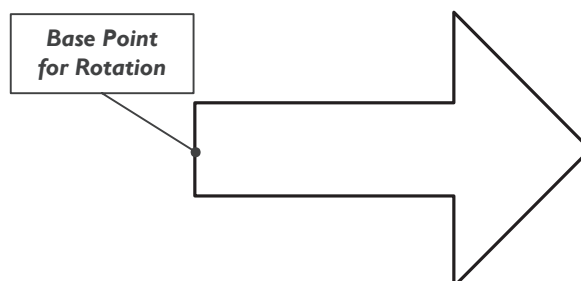


Left: Line before extending. **Right:** After extending both ends.

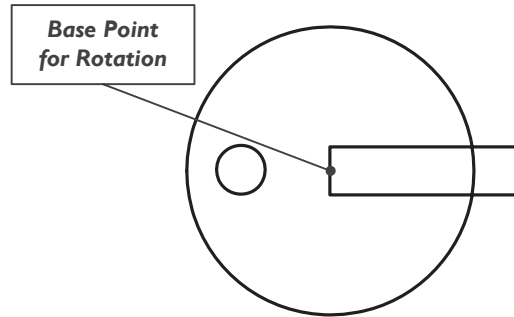
Extend the horizontal line to the vertical lines with the **EXTEND** command.

To extend the other end (remembering to choose both vertical lines as boundaries), respond to the repeating “Select object to extend:” prompt by selecting a point at the other end of the horizontal line.

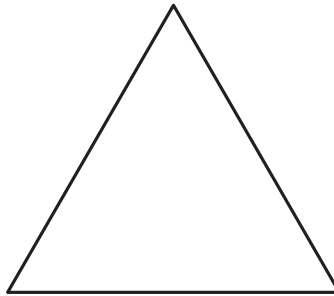
7. In this exercise, you rotate objects. Draw the arrow symbol illustrated by the figure. Using the **ROTATE** command, turn the arrow by 45 degrees about the center of the arrow’s base. Which object snap mode helps you find the midpoint of a line? Using grips editing, turn the arrow a further 90 degrees.



Draw a cam similar to the one illustrated by the figure.
Use Window object selection to rotate the entire cam by -45 degrees.



8. In this exercise, you resize objects.
With the **POLYGON** command, draw the triangle illustrated by the figure. The edge is 2 units long.
Use the **SCALE** command to double the size of the triangle.



Repeat the command, but this time use the **Reference** option again to double the size of the triangle.

Specify scale factor or [Reference]: **r**

Specify reference length <1>: **2**

Specify second point: **4**

Did AutoCAD resize all the lines making up the triangle?

9. In this exercise, you “stretch” a window to move it within the wall — without editing the wall.

From the CD, open the *stretch1.dwg* file, a drawing of a wall cross section.



Use the **STRETCH** command to move the window along the wall.

Which selection mode must you use?



Did the wall stretch?

10. From the CD, open the *stretch2.dwg* file, a drawing of a pencil.



Use the **STRETCH** command to make the pencil shorter.



11. In this exercise, you lengthen and shorten objects with a different command.

Draw an arc, using these parameters:

Start point **2,5**

Second point **0,3**

Endpoint **2,1.5**

Start the **LENGTHEN** command, and select the arc. What is its length and included angle?

Use the **DElta** option to add 2 units to the length.

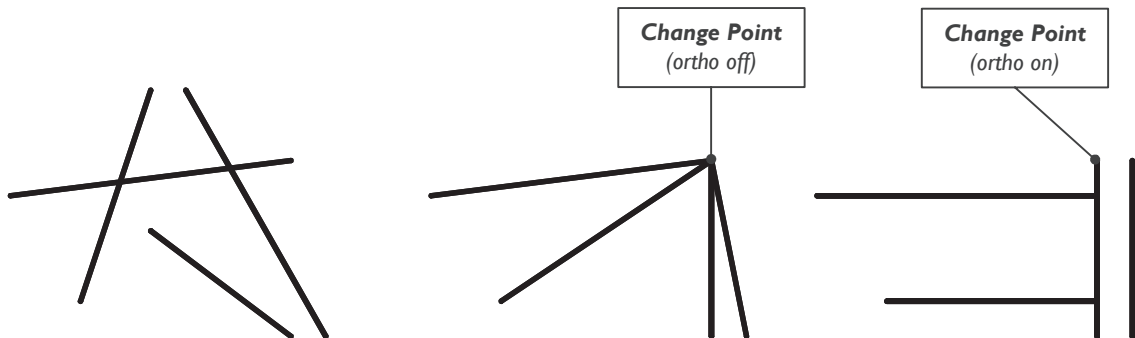
Use the **Total** option to change the arc to 5 inches. Did it become longer or shorter?

Use the **Percent** option to change the arc to 100%. Did the arc change?

12. In this exercise, you change objects.

Ensure ortho mode is turned off.

Draw four lines, randomly, as illustrated by the figure.



Use the **CHANGE** command to select the lines, and then use the **Change Point** option to give the lines a common endpoint.

Now use the **CHANGE** command's **Properties** option to change the lines to red.

13. In this exercise, you explode objects.

From the CD, open the *edit2.dwg* file, the drawing of the puzzle.

Select one of the puzzle pieces. What does the single grip tell you?

With the piece still selected, use the **EXPLODE** command. Does the piece look different?

Select the piece again. Has the number of grips changed? What does this tell you?

Use the **EXPLODE** command a second time. What message does AutoCAD give you?

Select a different, unexploded piece.

Start the **XPLODE** command, and specify a color of red. What happens to the puzzle piece?

14. In this exercise, you edit polylines.

From the CD, open the *offset.dwg* file, the drawing of the small town.

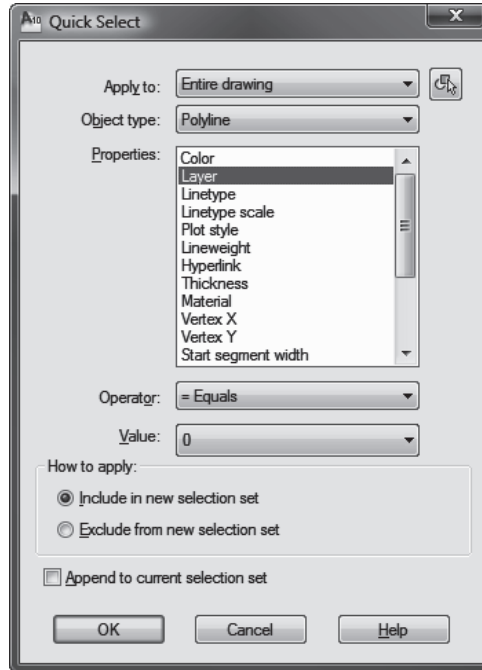
Select any of the polylines that define the edges of streets.

Use the **PEDIT** command's **Width** option to change the width to 0.1 units.

Exit the command, and start the **PROPERTIES** command.

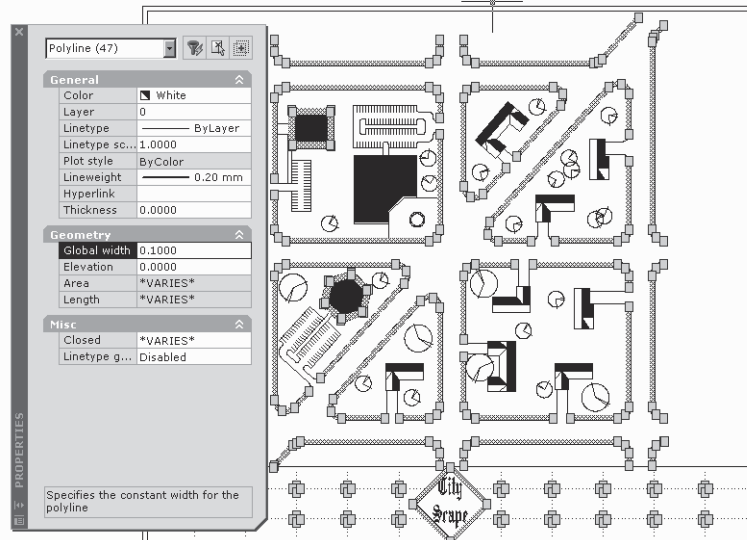
Select *all* the polylines in the drawing by clicking the **Quick Select**  button, and then entering these parameters:

Apply to	Entire drawing
Object type	Polyline
Properties	Layer
Operator	=
Value	0

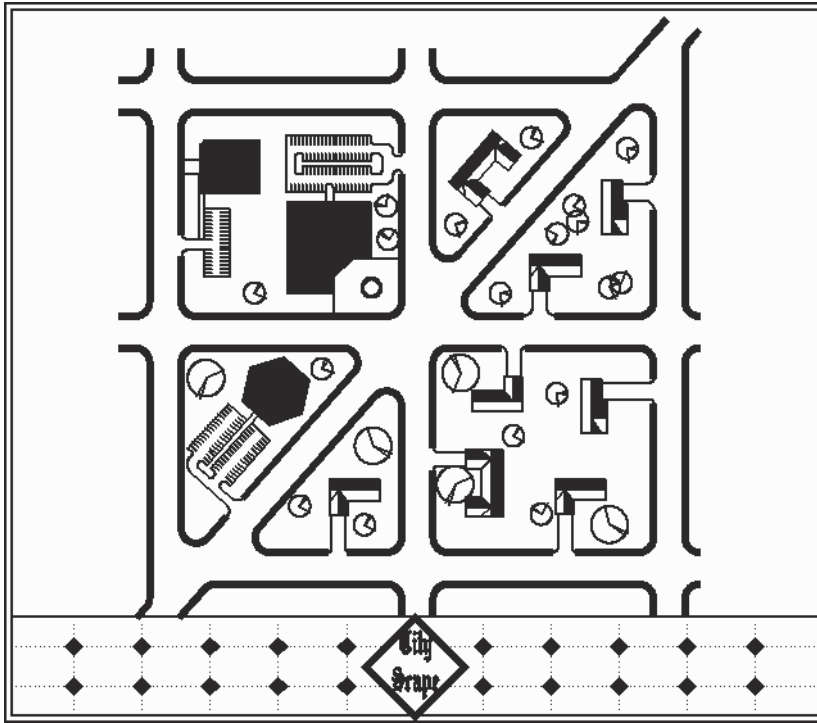


After you click **OK**, AutoCAD highlights all polylines.

In the Geometry section of the Properties palette, change **VARIES** to 0.1 units.



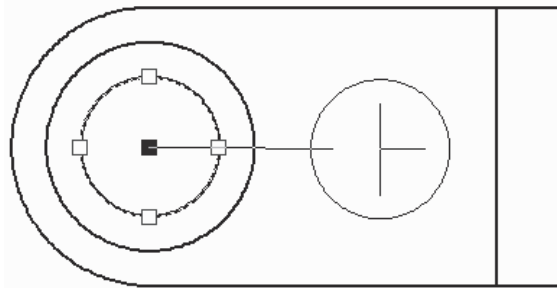
Press **ESC** to remove the grips and highlighting. The drawing should look like the figure below.



15. In this exercise, use grips to edit objects in the drawing.
From the CD, open the *17_35.dwg* file, a drawing of an angle.

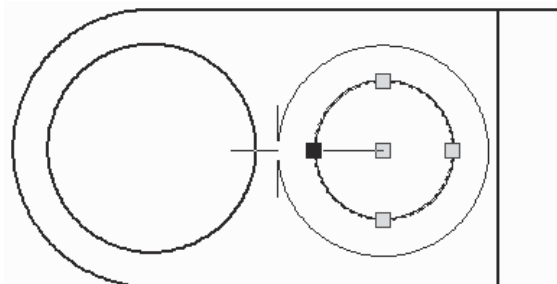
Select a circle, and then click on the center grip.

Drag the grip to another place in the drawing. Does the circle move?



Now click one of the four quadrant grips.

Drag the grip. Does the circle become larger?



Press **ESC** to remove the grips.

Select the entire drawing, and then click on any grip.

Press the spacebar to see **** MOVE **** on the command line.

Drag the grip. Do all parts of the drawing move?

Press the spacebar again to see **** ROTATE **** on the command line.

Drag the grip. Does the drawing rotate around the grip?

Enter **b** to access the base point option:

Specify base point: *(Pick another point.)*

Move the cursor. Does the drawing now rotate about the new point?

Press the spacebar again to see **** SCALE **** on the command line.

Drag the grip. Does the drawing change its size?

Press the spacebar again to see **** MIRROR **** on the command line.

Drag the grip. Do you see a mirrored copy?

16. Use the **LINE** command with direct distance entry to draw the object defined by the following relative coordinates:

Point 1: 0,0

Point 2: @3,0

Point 3: @0,1

Point 4: @-2,0

Point 5: @0,2

Point 6: @-1,0

Point 7: 0,0

17. Use the **PLINE** command with direct distance entry to draw the object defined by the following relative coordinates:

Point 1: 0,0

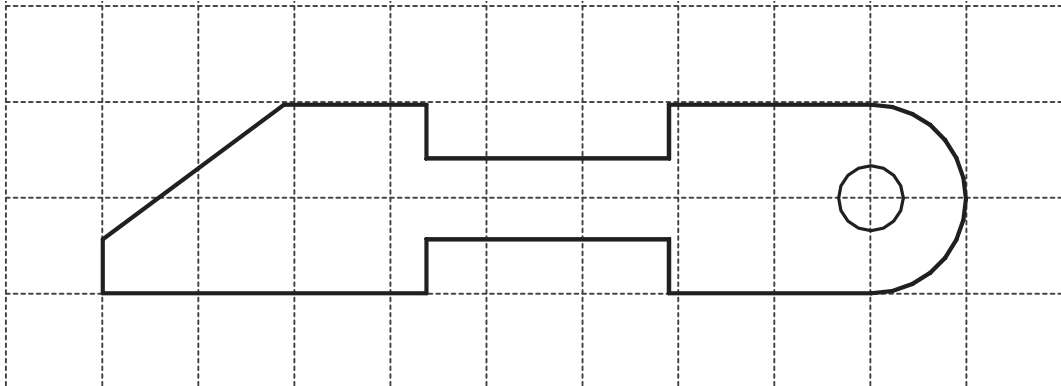
Point 2: @4<0

Point 3: @4<90

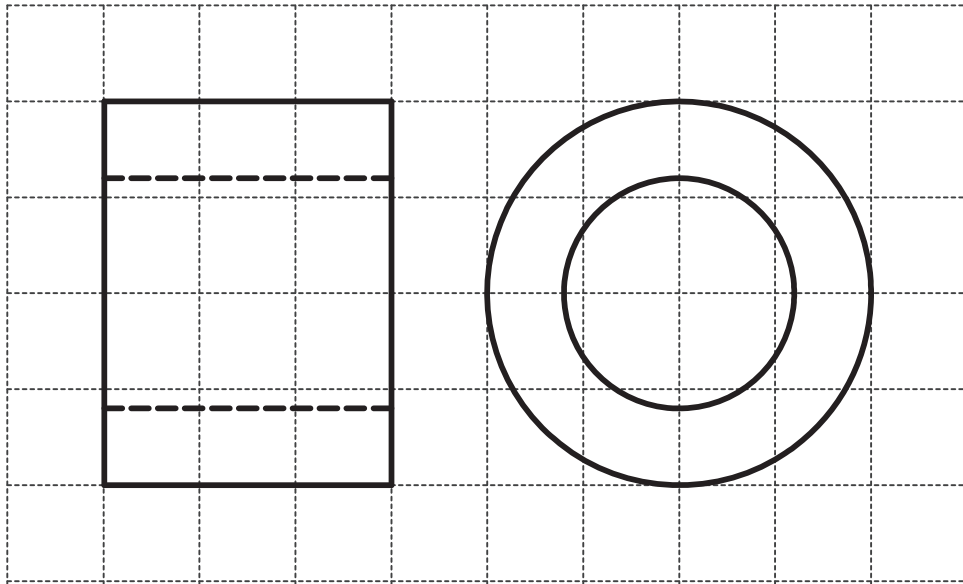
Point 4: @4<180

Point 4: @4<270

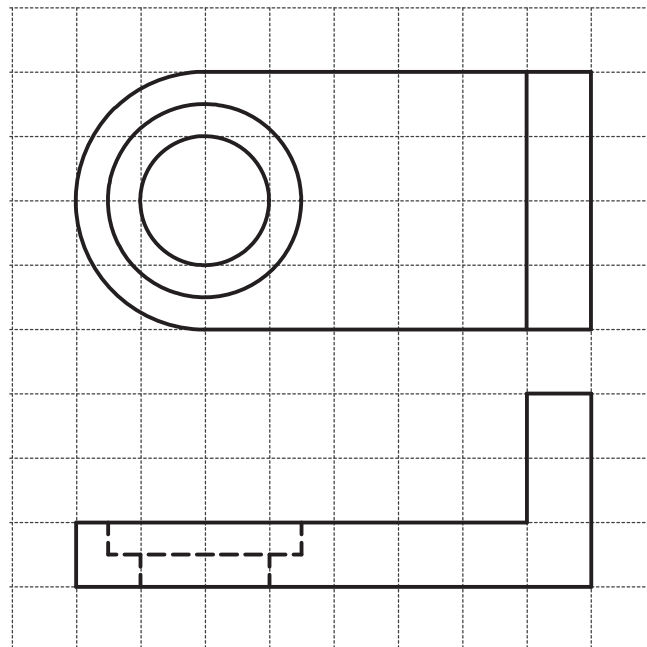
18. Use direct distance entry and tracking to draw the following fuse link. Each square represents one unit.



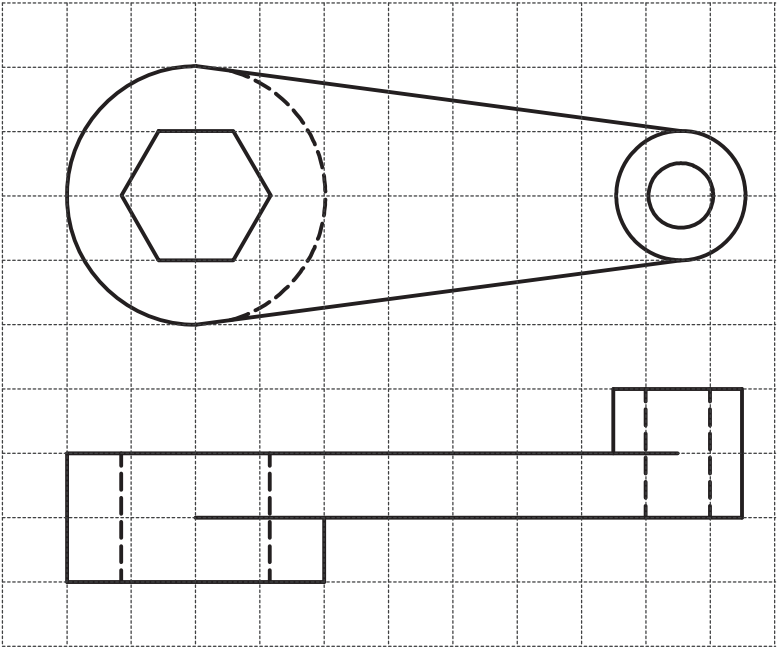
19. Use direct distance entry and tracking to draw the following cylinder. Each square represents 2 units.



20. Use direct distance entry and tracking to draw the following baseplate. Each square represents a half-unit.



21. Use direct distance entry and tracking to draw the following wrench. Each square represents one unit.



CHAPTER REVIEW

1. What is the difference between hot and cold grips?
2. When moving objects, must the base point be on the selected objects?
3. What objects are affected by the **BREAK** command?
4. In breaking objects, what happens if you do not enter **F** for selection of the first point?
5. Which commands restore objects just erased?
6. What happens when you explode blocks?
7. Describe what happens when you explode polylines.
8. What are *cutting edges*?
9. At what point are polylines trimmed?
10. What is the special requirement before circles can be trimmed?
11. Name the border line used by the **EXTEND** command.
12. Describe three ways to rotate objects.
 - a.
 - b.
 - c.
13. Can objects be resized differently in the x and y axes with the **SCALE** command?
14. What value changes objects to half their original size?
15. Describe the purpose of the **Copy** option of the **ROTATE** and **SCALE** commands.
16. Name the command that explicitly edits polylines?
17. Which parts of the polyline are the *vertices*?
18. How do you edit polyline vertices?
19. What types of vertex editing can you perform?
20. List the aliases for the following commands:
 - TRIM**
 - BREAK**
 - EXTEND**
 - PEDIT**
21. When can the **OOPS** command be used?
22. Is it possible to erase all objects in the drawing?
23. Can the **BREAK** command shorten objects?
24. What do circles become after you apply the **BREAK** command?
 - An ellipse?
 - Xlines?
25. Describe how the **LENGTHEN** command differs from the **TRIM** and **EXTEND** commands.
26. Can the **TRIM** command extend open objects?
 - If so, how?
27. Can the **EXTEND** command extend closed objects?
 - If so, how?
28. Do blocks and dimensions work as cutting edges?
 - Do hatches?
29. In what kind of drafting are the **TRIM** command's **Project** and **Edges** options used?
30. List the two things AutoCAD needs to know before extending objects:
 - a.
 - b.

31. Do blocks work as boundary edges?
Do hatches?
32. Can circles be extended?
33. Is it acceptable to select all objects as cutting edges?
34. Can arcs be lengthened with the **LENGTHEN** command?
Shortened?
35. Describe how the **LENGTHEN** command's **Total** and **Delta** options differ.
36. When does the **STRETCH** command only move objects?
37. Can objects be moved using grips editing?
38. What does the **ROTATE** command rotate objects about?
39. Where does 0 degrees point, by default?
Can the direction be changed?
If so, how?
40. In which direction does AutoCAD measure negative angles, by default?
41. What three pieces of information does AutoCAD need to know before resizing objects?
 - a.
 - b.
 - c.
42. Describe how the **CHANGE** command's **Change Point** option changes lines with:
Ortho mode turned off:
Ortho mode turned on:
43. What is the difference between the **EXPLODE** and **XPLODE** commands?
44. How are arcs in nonuniformly scaled blocks exploded?
45. Under what condition must blocks be exploded more than once?
46. What are polylines exploded into?
47. Can you specify the color of exploded objects?
If so, how?
48. Describe how AutoCAD reacts when you select lines and arcs with the **PEDIT** command?
49. Can lines and arcs that don't touch be turned into a single polyline?
If so, how?
50. During polyline vertex editing, which keystrokes move the x marker from vertex to vertex?
51. How does the **Straighten** option affect a polyline?
52. What is the difference between the **Fit** and **Spline** options of the **PEDIT** command?
53. Are the splines created by **PEDIT** true splines?
54. Explain why the **Ltype gen** option is important?
57. What does the padlock icon indicate?
58. What are the default colors for the following grips:
Hot:
Cold:
Hover:
59. Name three editing operations you can perform with grips.
60. How do you deselect gripped objects?
61. What is the primary difference between *tracking* and *direct distance entry*?
62. What is the primary purpose of grips editing?

63. Where are grips located on lines?
On circles?
On blocks?
64. Using grips, can more than one object be stretched at a time?
65. Describe the purpose of the **Base point** option in grips editing.
66. What happens when you drag a circle by its center point grip?
What happens when you drag a circle by its quad grip?
67. Can direct distance entry be used with editing commands?
68. What is the alias for tracking?
69. Name the mode that AutoCAD automatically turns on when you enter tracking mode?
70. Other than the standard six grips editing commands, name three other editing commands that work with gripped objects:
 - a.
 - b.
 - c.
71. Can you use the **BLOCK** command with gripped objects?
72. How would you find the midpoint between two other points?

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Text and Dimensions

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CHAPTER 10

Placing and Editing Text

Text is used in many areas of drawings: title blocks that identify drawings, callouts that identify parts, bills of material that list parts, and paragraphs of text that warn and explain. Traditionally, placing text by hand in a paper drawing was laborious; drafters drew each letter individually. In contrast, placing text in an AutoCAD drawing requires little effort. This chapter shows how to place text in drawings, and then how to change the text and its properties.

The commands are:

TEXT places text in drawings, a line at a time.

MTEXT places paragraphs of formatted text.

TEXTTOFRONT displays text in front of other objects.

QLEADER and **MLEADER** place callouts in drawings.

TEXTEDIT edits text (renamed in AutoCAD 2010).

STYLE define named text styles based on fonts.

MLEADERSTYLE defines named multiline leader styles.

SPELL checks the drawing for unfamiliar words.

FIND searches and optionally replaces text.

SCALETEXT and **JUSTIFYTEXT** change the size and justification of text.

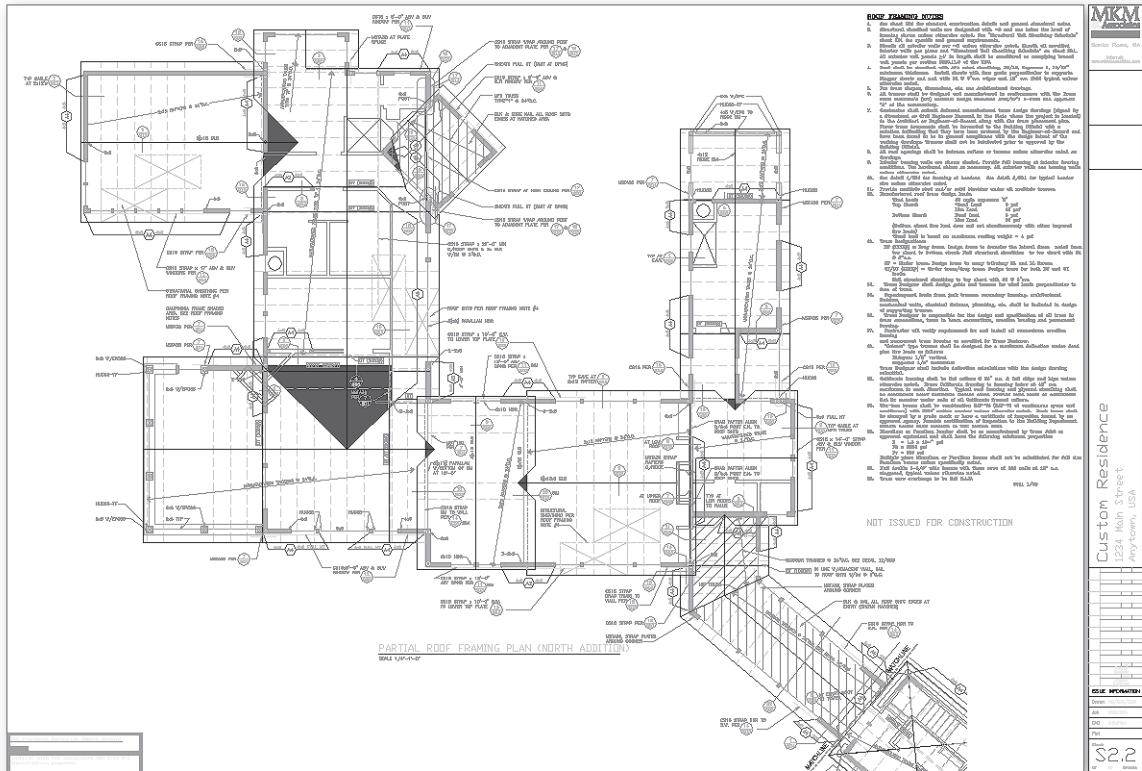
Annotation property displays correctly-scaled text in model space.

NEW TO AUTOCAD 2010 IN THIS CHAPTER

- **DDEDIT** has been renamed **TEXTEDIT**.
- **MTEXT**, **FIND** and **MLEADER** commands have new features.
- **SPELL** dialog box adds an Undo button.
- **MLEADERSTYLE** now specifies scale factors for block annotations, and provides direct access to the **STYLE** command's dialog box.

TEXT IN DRAWINGS

The figure below illustrates many uses of text in drawings. Can you spot the general notes, title block, section view numbers, leaders, and dimensions? Methods for creating these are described in this and following chapters.



The most important requirement of text is that it be clear and concise. In the days of hand drafting, neophyte drafters learned how to print letters and numbers neatly first, and only then how to draw geometry. Today, AutoCAD does the lettering for us, but the same rule applies: clear text is crucial to understanding drawings. (AutoCAD's default of 0.2 units is left over from the days of dot-matrix printers.)

Here are some guidelines for placing text:

- **Headings** are 3/16-inch high.
- **Note text** is 1/8-inch high.
- **Text** is left-justified (each line of text aligned at its left edge).

You can calculate the size of the text manually, or let AutoCAD do it automatically through annotative scaling.

The use of text in drafting is governed by standards. Many North American industries use the American National Standards Institute (ANSI) style of letters and numbers. European companies use ISO lettering (International Organization for Standardization), while other companies and countries set standards of their own.

FONTS

In drawings, text is created through fonts, and AutoCAD is versatile at displaying fonts. They can be stretched, compressed, obliqued (slanted), and mirrored (reversed); in some cases, the text can even be placed vertically. You can apply colors, lineweights, and plot styles to text — though not linetypes.

The default font in every new AutoCAD drawing is the common Arial font. Other clear fonts often used by drafters are called “Simplex” and “RomanS.”

Arial font Simplex font

Left: AutoCAD’s default style, the Arial font.

Right: AutoCAD’s Simplex font preferred by many drafters.

Fonts are the “design” of text letters. Caslon, for example, is the name of the font used for the text in this paragraph, while GillSans is used for the headings and tutorial text in this book. Arial, TimesRoman, and Courier are among the most commonly-used fonts in documents.

Fonts are defined by files that come in several formats. There is AutoCAD’s own SHX format (.shx files) that Autodesk developed many years ago. But now it is more common to use TrueType fonts (.ttf files) in drawings; these were designed by Apple and are found on all computers running Windows. A third format, PFB (.pfb files), was designed by Adobe and is typically used for typesetting; it can be used in AutoCAD after conversion to SHX format with the CONVERT command.

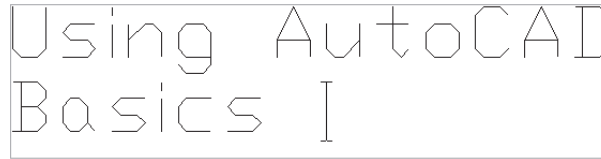
AutoCAD has two commands for placing text (TEXT and MTEXT) and a third for defining text styles (STYLE). The TEXT command places lines of text in drawings, while MTEXT places paragraphs of text. TEXT is like the minimalist Notepad text editor, while MTEXT is more akin to the full-featured Word word processing application.



The TEXT command places lines of text in drawings.

TUTORIAL: PLACING LINES OF TEXT

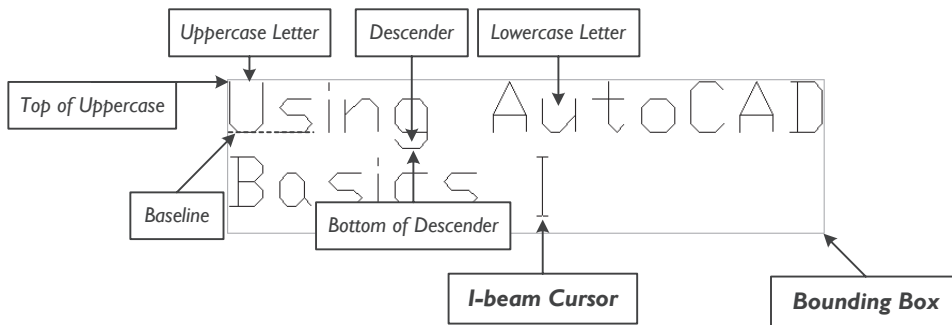
- To place lines of text in drawings, start the **TEXT** command:
 - In the ribbon’s Home tab, pick the **Single Line Text** button in the Annotation panel.
 - At the ‘Command:’ prompt, enter the **TEXT** command:
Command: **text** (Press ENTER.)
 - Alternatively, enter the **dt** or **dtext** (the old name for this command) aliases at the ‘Command:’ prompt.
- In all cases, AutoCAD reports the current text style, height, and annotation scale settings, and then prompts you for the location of the start point:
Current text style: “Standard” Text height: 0.2000 Annotative: No
Specify start point of text or [Justify/Style]: (Pick a point, or specify x, y coordinates.)
- Specify the height of the text and its rotation angle.
Specify height <0.2000>: **0.75**
Specify rotation angle of text <0>: (Press ENTER to accept the default.)
In most cases, the rotation angle is 0 degrees; sometimes, however, you may need to place text at an angle, such as at 45 or 90 degrees.
- Enter a line of text, and then press **ENTER**.
Using AutoCAD (Press ENTER.)
- After you press **ENTER**, the cursor jumps to the start of the next line.
When you enter additional lines of text, AutoCAD places them under the previous line.

Basics (Press ENTER.)


6. To exit the command, press **ENTER** a second time:
Enter text: (Press ENTER to exit the command.)

If you cancel the **TEXT** command by pressing **ESC**, the current line of text is erased; so, remember to press **ENTER** instead.

As you type text, AutoCAD displays the *I-beam cursor*, called that because it looks like the letter I. The cursor has a pair of short horizontal lines that represent the top of uppercase letters and the bottom of descenders. (A *descender* is the part of lowercase letters that hangs below the baseline — for instance, the letters g, q, and y have descenders.)

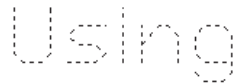


The text you enter is surrounded by a gray *bounding box*, which shows the extent of the text.

If you make a mistake, press **BACKSPACE** to erase one character at a time; alternatively, highlight a group of characters to change or erase them.

During the **TEXT** command, you'll find that ribbon selections, command options, and other functions are "locked out." Only keyboard entry is permitted during this command.

The next time you use the **TEXT** command, the last text string you entered is *highlighted* (shown in dashed lines):



AutoCAD does this for a good reason: for placing additional lines of text. When you press **ENTER** at the 'Specify start point of text:' prompt, the I-beam cursor is placed on the next line after the previous text string — as though you had never exited the command:

Command: (Press ENTER to repeat the **TEXT** command.)

TEXT Current text style: "Standard" Text height: 2.0000

Specify start point of text or [Justify/Style]: (Press ENTER to continue text below the last line.)

AutoCAD skips the height and rotation prompts, because it assumes you want them to be the same as before.

PLACING TEXT ALL OVER DRAWINGS

The **TEXT** command is useful for placing text quickly in many places on the drawing: simply click the cursor in another location, and then continue typing. This trick is handy for filling out title sheets and other text intensive jobs.

AutoCAD calls these pockets of text “blocks” (not to be confused with blocks made of objects.) During the **TEXT** command, you can move between text blocks by pressing the **TAB** key. Press **SHIFT+TAB** key to return to a previous text block; press **TAB** again to jump ahead to the next text block.

To split a line of text in two, use the cursor arrow keys (or the mouse) to locate the cursor at the break, and then press **ENTER**.

PLACING TEXT: ADDITIONAL METHODS

The **TEXT** command allows you to control the justification and style of the text, as listed by the options below.

- **Justify** specifies a text justification mode.
- **Style** selects a predefined text style.
- **Height** specifies the height of the text.
- **Rotation angle** specifies the angle at which the line of text is rotated.
- **Control Codes** add formatting and special characters.
- **DTEXTED** system variable controls the look and feel of the command.

Let’s look at each.

Justify

When you respond to the prompt ‘Specify start point of text’ by picking a point or typing x, y coordinates, this becomes the starting point for the text string. If you want a different text alignment, then enter the **Justify** option. (This is also known as the “insertion point.”)

This is an
example
of left
justified
text

This is an
example of
center
justified
text

This is an
example of
right
justified
text

Left: Left justified text.
Center: Center justified text.
Right: Right justified text.

The default justification is *left-justified*, which means that all lines of text start at the left edge. Other common justifications include center-aligned (often used in title blocks) and right-justified, used for text placed to the left of leader lines.

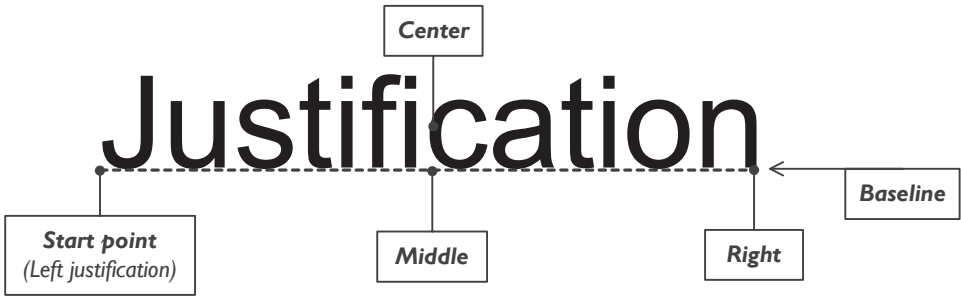
To see the list of all text justification modes, respond with a **J** (short for “justify”) at the ‘Specify start point of text or [Justify/Style]:’ prompt:

Specify start point of text or [Justify/Style]: j

AutoCAD asks you to enter an alignment option:

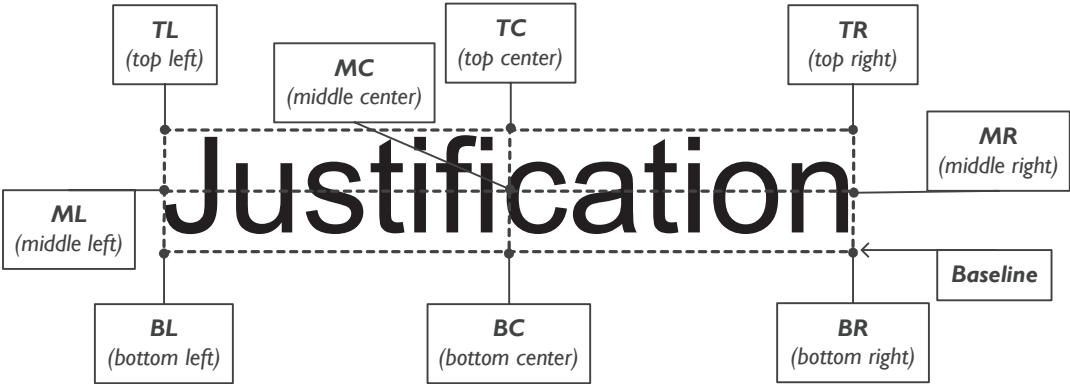
Enter an option [Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR]:

The list of justification modes is long, because text can be justified (aligned) both vertically and horizontally. AutoCAD uses one- and two-letter abbreviations to designate each alignment option. (History: The one-letter options were present in the original AutoCAD 25 years ago; the two-letter options were added more recently.) The one-letter justification modes are most commonly used:



Justify	One-letter Options
Start point	Default; equivalent to left and BL (see below).
A	Align; fits text between two points, and adjusting height appropriately.
C	Center; equivalent to BC.
F	Fit; fits text between two point at a specific height.
M	Middle; equivalent to MC.
R	Right; equivalent to BR.

When it comes to two-letter options, the first letter describes the vertical alignment; the second, the horizontal alignment. For example, TL is top-left. The figures illustrate the alignment modes listed in the tables.



Justify	Two-letter Options
TL	Top left.
TC	Top center.
TR	Top right.
ML	Middle left.
MC	Middle center.
MR	Middle right.
BL	Bottom left.
BC	Bottom center.
BR	Bottom right.



Note If you know the alignment you want, it is not necessary to enter the **J** option; simply enter the one- or two-letter abbreviation for the alignment. For example, to right-justify text:

Specify start point of text or [Justify/Style]: r

Align

The **Align** option requires you to select two points, and then it fits the text between them. AutoCAD automatically adjusts the text height so that the baseline of the text fits perfectly between the two points. Note that the two points can be placed at any angle in relation to each other.



Command: **text**

Current text style: "Standard" Text height: 0.2000

Specify start point of text or [Justify/Style]: **a**

Specify first endpoint of text baseline: (Pick point 1.)

Specify second endpoint of text baseline: (Pick point 2.)

Enter text: (Type text.)

Enter text: (Press ENTER to exit command.)

There are no "height" and "rotation angle" prompts, because the pick points determine the angle of the text. AutoCAD determines the height.

When you pick two points in reverse order (right to left), AutoCAD draws the text upside down, as illustrated below.



Fit

The **Fit** option is similar to Aligned: it prompts you for two points between which to place the text. But it also prompts for the text height. AutoCAD forces the text to fit between the two points, and draws the text at the height you specify.



Examples of text fitted between two points, but assigned different heights.

Specify start point of text or [Justify/Style]: **f**

Specify first endpoint of text baseline: (Pick point 1.)

Specify second endpoint of text baseline: (Pick point 2.)

Specify height <0.2>: (Enter a height.)

Enter text: (Type text.)

Enter text: (Press ENTER to exit command.)

The text is squashed or stretched horizontally to fit between the two points, as illustrated above. Fitted text is often used in constrained areas, for example when labeling small closets. ("Or,"

suggests, the technical editor, “You could come out of the closet and put the label in the hall.”) When you pick the second point to the left of the first, AutoCAD draws the text upside down.

Style

The **Style** option selects a predefined text style. *You must first create a text style before you can use it.* The exceptions are the Standard and Annotative styles, which are included in every new drawing.

Styles (covered in detail later in this chapter) define the many parameters of text, such as the font it uses and whether it is shown boldface or has a predefined height.



Note The **Style** option of the **TEXT** command and the **STYLE** command are not the same. The **Style** option selects styles, while the **STYLE** command creates styles.

Height

The **Height** option determines the height of the text, which can be any distance — as tall as the orbit of Pluto, if need be. (The technical editor corrects the author by noting that text can in fact be taller than the orbit of Pluto. The tallest text can be as tall as 9.49×10^{94} units — considerably taller than the known universe. “Tower of Bable-size font,” remarks the copy editor.) The default is 0.2 units.

Specify height <0.2000>: (Enter a height such as **17.5**, or press **ENTER**.)

Once the text height is set, it remains the default until you change it.

There are two conditions, however, under which the ‘Specify height’ prompt doesn’t show up. One is when the height is predefined by the style; another is during the Align option.

Determining Text Height

Like hatch patterns and linetypes, text must be scaled appropriately for the drawing. If you were to sketch a picture of your house on a piece of paper, you would draw it small enough to fit the paper. (There *might* be a sheet of paper big enough to draw your house full size, but it would be hard to find!) Because *scale* is so important for text, a review may be in order.

In a sketch of my 50-foot house, I drew it 4 1/4" long. My drawing of the house is 140 times smaller than its actual size. The drawing scale is 1:140.

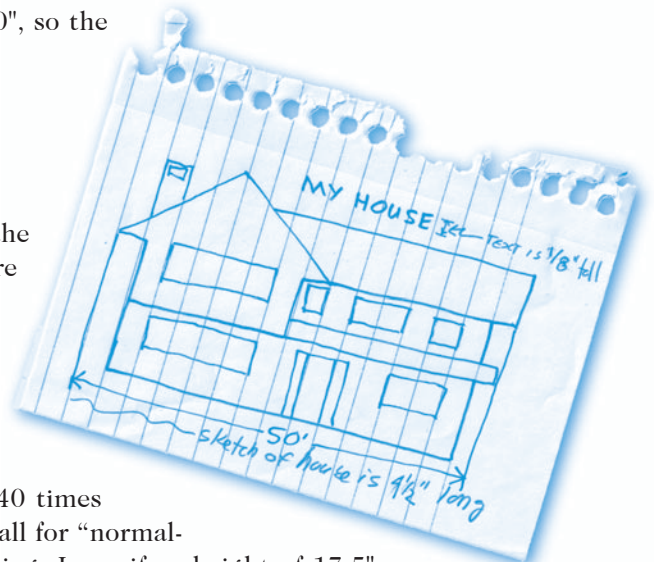
Here is how I arrived at 1:140. Convert the feet and inches to a common unit of inches. That makes 50' equal to 600", so the scale works out to be:

$$\frac{4.25"}{600"} = 1:140 \text{ (roughly)}$$

The 1:140 is the scale factor. The text I wrote on the paper, however, is *full size*, which happens to measure 1/8" tall in my sketch.

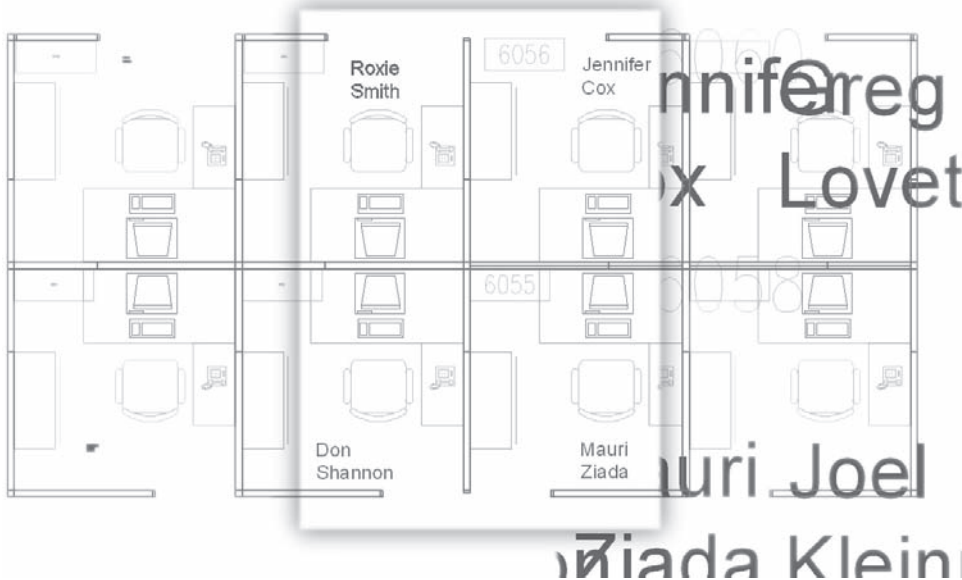
In AutoCAD, scaling is done in reverse order from hand sketches. Instead of drawing the house smaller, I draw it full size: the 50-foot house is drawn 50 feet long in AutoCAD, at a scale of 1:1.

The text cannot be drawn at 1:1 — it must to be 140 times larger. The standard in drafting is to draw text 1/8" tall for “normal-sized” text. When I place the text in my house drawing, I specify a height of 17.5"



tall ($\frac{1}{8}$ " x 140). This may seem to you much too tall (nearly one-and-a half feet high!) but trust me: when plotted, it will look exactly right.

The figure below illustrates text placed at several heights — some too small, some just right, and some too large. Whether it is right or wrong depends on the plotted size.



Left: Text height of 1" being too small.
Center: Text height of 6" being just right.
Right: Text height of 2' being too tall.

Annotative Scaling

There are two approaches to solving the text height problem. One is to work out the text size, as described above, and then enter it when the TEXT command prompts you:

Specify height <0.2000>: **17.5**

The better solution is to use *annotative scaling*, in which AutoCAD determines the text height automatically. You tell it the plotted height (a.k.a. paper height), and AutoCAD sizes the text based on the viewport's scale factor.

Annotative scaling is available only when it is turned on with the STYLE or PROPERTY commands. Anything affected by annotative scaling has the  icon near it, the end view of an engineer's scale ruler. (All new drawings contain the Annotative text style, although Standard continues to be the default.) When annotative scaling is turned on, the TEXT command changes the wording of the height prompt by adding the word "paper":

Specify **paper** height <0.2000>: **0.125**

You enter the height at which you wish the text to appear plotted on paper. Thus, text $\frac{1}{8}$ " (0.125 units) tall is plotted $\frac{1}{8}$ " tall, no matter the scale of the viewport. This eliminates the need to figure out the scale factor. But there are some nuances involved in annotative scaling, as detailed near the end of this chapter.

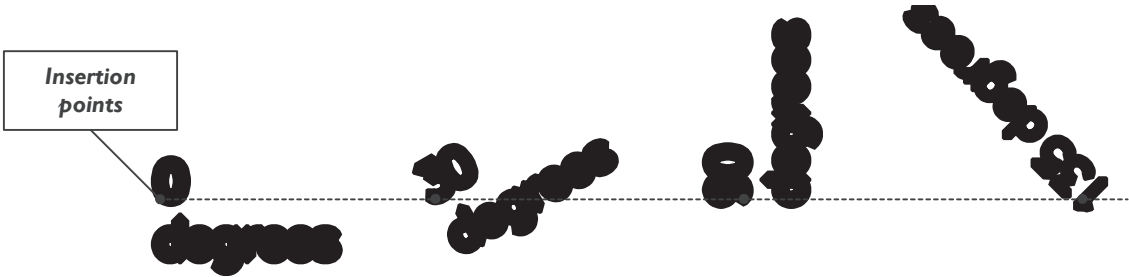
Rotation Angle

Lines of text can be placed at any angle in drawings. You specify the angle when you are prompted:

Specify rotation angle of text <0>: (*Enter an angle, or pick two points.*)

Press ENTER to accept the default angle, 0 degrees in this case. The text is rotated about the *insertion point*, whose location varies depending on the justification mode.

Once the text angle is set, the angle remains the default until you change it. The figure illustrates text placed at several angles.



The angle is affected by the ANGBASE and ANGDIR system variables, as are other angles in AutoCAD.

Control Codes

AutoCAD allows you to add *metacharacters* (also known as “control codes”) to text. Metacharacters mean something other than themselves. (As a professor of English literature, the copy editor notes that *all* text means something other than itself. We will sidestep the philosophical discussion over what is meta and what isn’t.) Control codes specify underlined text and symbols, such as the degree and plus/minus.

While the TEXT command is running, you can type codes that start with two percent characters: %%. The following table lists the available codes and their meaning:

Control Code	Meaning	Sample
%%c	Diameter symbol	Ø
%%d	Degree symbol	°
%%o	Start and stop overline	<u>Text</u>
%%% or %	Percent symbol	%
%%p	Plus-minus symbol	±
%%u	Start and stop underlining	<u>Text</u>
%%nnn	Any ASCII character <i>nnn</i>	%%126 = ~

Let’s look at an example with the following text string:

```
Command: text
Current text style: "Standard" Text height: .02 Annotative: No
Specify start point of text or [Justify/Style]: (Pick a point.)
Specify height <0.02>: (Press ENTER.)
Specify rotation angle of text <0>: (Press ENTER.)
Enter text: If the piece is fired at 400%%dF for %%utwenty%%u hours,
Enter text: it will achieve %%p95%% strength.
Enter text: (Press ENTER.)
```

Codes are initially displayed instead of the effect; once you begin typing the next text, the code disappears, replaced by the formatting:

If the piece is fired at 400° for twenty hours,
it will achieve ±95% strength.

Notice that you turn on and off the underscore (underlining) by typing %%u twice — once to turn on, and once to turn off. If you forget to turn it off, AutoCAD does it for you at the end of the line.

You can “overlap” symbols, for instance, using the degree symbol between the underscore symbols. To get this **Fire at 400°F.**, enter this:

%%uFire at 400%%dF%%u.

ASCII Characters

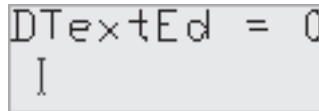
ASCII characters refer to the ASCII (short for “American Standard Code for Information Interchange”) character set, which assigns number codes to symbols.

To use a symbol in the text, enter two percent signs (%%) followed by the ASCII character code. For example, to place a tilde (~) in your text, enter %%126.

DTextEd

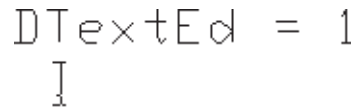
The **TEXT** command has changed over the years, and the **DTEXTED** system variable controls the changes. (The sysvar is named after the old **DTEXT** command, short for “dynamic text.”)

- 2** — displays the Inplace Text Editor for writing and editing text; you can click to continue text elsewhere in the drawing. Default action of **TEXT** since AutoCAD 2007.



DTextEd = 0
I

- 1** — displays the ‘Enter text.’ prompt at the command line; you can click elsewhere in the drawing to start new text strings. Simulates the action of **TEXT** in AutoCAD 2005 and earlier.



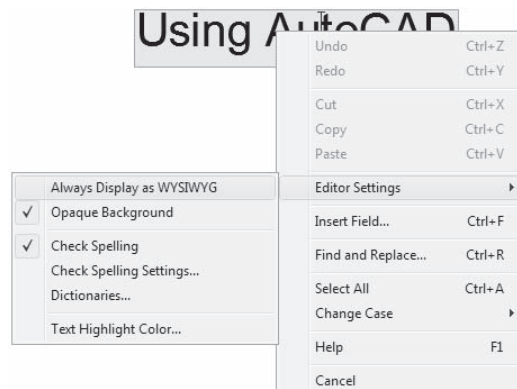
DTextEd = 1
I

- 0** — displays the Inplace Text Editor for writing and editing text; you cannot click to continue text elsewhere. Simulates the action of **TEXT** in AutoCAD 2006.

I recommend you leave this system variable set to 2.

TEXT SHORTCUT MENU

The **TEXT** command sports a shortcut menu with useful options. To see the menu, right-click text during the command. (This works when **DTEXTED** is set to 0 or 2; when set to 1, the shortcut menu doesn’t appear.)



The significant options are described next.

Always Display as WYSIWYG

The **Always Display as WYSIWYG** option edits text in-place. (WYSIWYG is short for “what you see is what you get.”) When this option is on, and text is at some unusual angle, the text editor edits it that way, as illustrated below.



Opaque Background

The **Opaque Background** option fills the bounding box with gray (default = off). The color helps distinguish the text input area, and appears only while editing; when you exit the command, the color disappears.

Using AutoCAD

Using AutoCAD

Left: No background color.

Right: Opaque background..

Check Spelling

The **Check Spelling** option toggles real-time spell checking. As you enter or edit text, AutoCAD checks the spelling of each word against its list of correctly-spelled words. When it finds a word missing from the list, AutoCAD underlines the word with a red dashed line.

Using OttoCAD

Correct the spelling of the word. Alternatively, right-click to see a list of alternative spellings.



Choose **More Suggestions** for a sub-menu filled with additional words. Or, choose **Add to Dictionary** when you want AutoCAD to recognize the word in the future.

Check Spelling Options and Dictionaries

The **Check Spelling Options** items displays the Check Spelling Settings dialog box, while the **Dictionaries** option displays the Dictionaries dialog box. Both are the same as found in the Spell dialog box, described later in this chapter.

Text Highlight Color

The **Text Highlight Color** option specifies the color of highlighted text (default = bluejeans blue). This option brings up the Select Color dialog box, from which you choose another color. In the figure below, “AutoCAD” is shown highlighted.



Insert Field

The **Insert Field** option displays the Field dialog box. This allows you to add automatic text, as described fully in Chapter 17, “Tables and Fields.” Examples of field text include the current date and time, the file name of the drawing, and the plot scale.



Field text is identified by a dark gray background. When the field looks like “----”, it means the text hasn’t been determined yet, such as the name of the drawing not yet saved. When “####” appears, it means values have not yet been assigned.

Update Field

After the field is placed in the text, the shortcut menu shows two additional options: Update Field and Convert Field to Text.

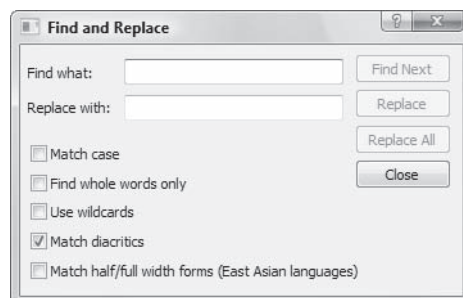
The **Update Field** option forces AutoCAD to update the value of fields. Normally, AutoCAD waits to update fields until certain events occur, such as when a file is opened, saved, plotted or regenerated; or when the ETRANSMIT command is used. This option update fields now, instead of later.

Convert Field to Text

The **Convert Field to Text** option does exactly that: the field is converted to static text. This is useful when you no longer want AutoCAD to update fields.

Find and Replace

The **Find and Replace** option displays a simple dialog box for finding (and optionally replacing) text.



I am not sure of the utility of this option, because the **TEXT** command is meant for small amounts of text, and so finding a specific word is not a problem. A better method is to use the **FIND** command, which searches for any kind of text in the entire drawing.

Select All

The **Select All** option selects all the text in the bounding box.

Change Case

The **Change Case** option changes text to all UPPERCASE or all lowercase. The change applies to selected text only, and it does not apply at all to field text.



Note The **TEXT** command is efficient for quickly placing strings of text in the drawing. (*Strings* is another way of saying “lines of text.”) **MTEXT** is best for paragraphs of text and text that contains formatting, such as **boldface** and **color**.

A MTEXT

The **MTEXT** command places paragraphs of text in drawings, and provides many formatting options.

While **TEXT** places text in the drawing one line at a time, the **MTEXT** command (short for “multiline text”) creates multiple paragraphs of text. It fits the text into an invisible boundary that lets you easily fit text to specific areas or to change margin widths. (The boundary is not printed or plotted.)

MTEXT permits more text enhancements than **TEXT**, such as varying heights and colors, and stacking fractions. The figure below illustrates paragraph text:

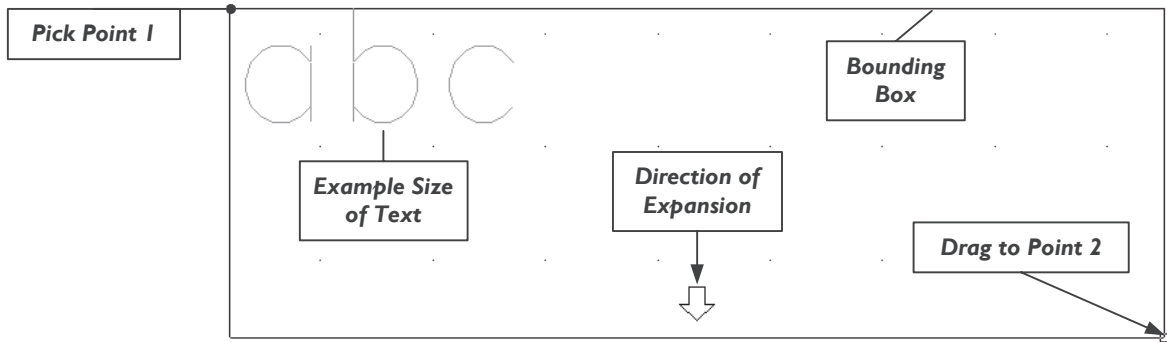
ROOF FRAMING NOTES

1. See sheet SD1 for standard construction details and general structural notes.
2. Structural sheathed walls are designated with ~G and are below the level of framing shown unless otherwise noted. See “Structural Wall Sheathing Schedule” sheet SD1 for specific and general requirements.
3. Sheath all exterior walls per ~R unless otherwise noted. Sheath all specified interior walls per plans and “Structural Wall Sheathing Schedule” on sheet SD1. All exterior wall panels $\geq 4'$ in length shall be considered as complying braced wall panels per section 2320.11.3 of the UBC.
4. Roof shall be sheathed with APA rated sheathing, 32/16, Exposure 1, 15/32” minimum thickness. Install sheets with face grain perpendicular to supports. Stagger sheets and nail with 8d @ 6” o.c. edges and 12” o.c. field typical unless otherwise noted.
5. For truss shapes, dimensions, etc. see Architectural drawings.
6. All trusses shall be designed and manufactured in conformance with the Truss

TUTORIAL: PLACING PARAGRAPH TEXT

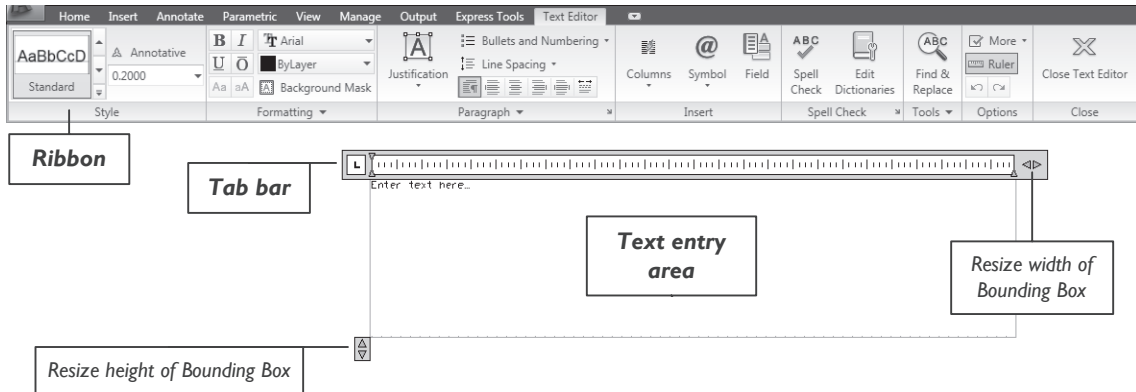
1. To place paragraphs of text in drawings, start the **MTEXT** command:
 - In the ribbon’s Home tab, pick the **Multiline Text** button from the Annotation panel.
 - At the ‘Command:’ prompt, enter the **MTEXT** command:
Command: **mtext** (Press ENTER.)
 - Alternatively, enter the aliases **t** or **mt** at the ‘Command:’ prompt.
2. In all cases, AutoCAD reports the current settings, and then prompts you to pick the two corners of the bounding box :
Current text style: "Standard" Text height: 0.2 Annotative: No
Specify first corner: (Pick point 1.)
Specify opposite corner or [Height/Justify/Line
spacing/Rotation/Style/Width/Columns]: (Pick point 2.)

The bounding box specifies the space in which the text should fit. Don’t worry about the box being exactly the right size; you can always make it bigger or smaller later. In any case, AutoCAD automatically makes it longer as text is added; the direction that the bounding box expands is indicated by direction of the arrow icon you see in the box.



(You can use the **MTJIGSTRING** system variable to change the preview text, shown by “abc” above.)

3. The text editor appears. It consists of two parts: the ribbon for formatting the text and a separate text entry area topped by a tab bar for setting margins and tabs.



4. Type text into the text entry area.
5. To format the text, you first *highlight* it: drag the cursor over the text you want formatted. Then, from the ribbon toolbar, select a format option.
6. When done, click **OK**. Notice that AutoCAD places the text in the drawing. To exit the mtext editor without saving changes, press **ESC**.



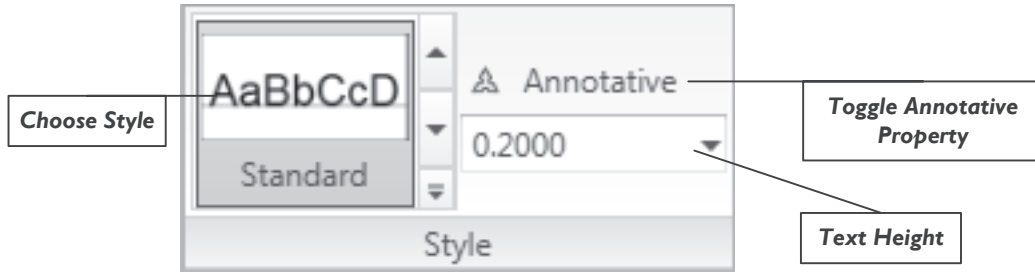
Note To insert the Euro symbol, turn on **NUMLOCK**, hold down the **ALT** key, and then type **0128** on your keyboard's numeric keypad. The € (Euro currency) symbol has been included with all AutoCAD fonts since Release 2000.

RIBBON MTEXT TAB

The MText tab of the ribbon allows you to apply styles to text; more important, it lets you override styles with other properties, such as font and color. We look below at each panel found in the tab.

Style Panel

Select a style name from the Style droplist. Only text styles defined in the drawing are listed here; you create additional styles with the **STYLE** command. The style applies to all text in the bounding box; you cannot selectively apply styles. You can, however, override styles with font, text height, and other properties provided in the Text Formatting toolbar.



Annotative Property

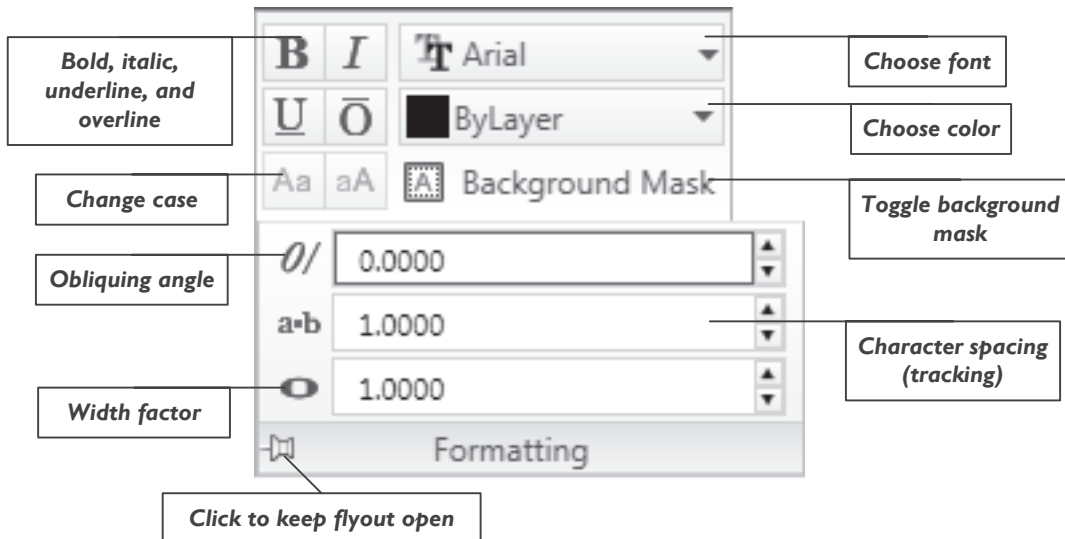
Click the  annotative button to change the font scaling to annotative. You learn more about this option near the end of this chapter.

Text Height

If you want some (or all) of the text at different heights, highlight the text, and then select a height from the Text Height dropdown.

If the height you need isn't on the list, simply type the measurement into the droplist, and then press ENTER. AutoCAD adds the height to the list.

Formatting Panel



Bold, Italic, Underline, and Overline

To **boldface** text, highlight the text, and then choose the **B** button. Choose the button a second time to “unboldface” the text. Pressing CTRL+B performs the same function.

When the **B** on the button is gray, the current font cannot be boldfaced — usually a problem with certain AutoCAD fonts.

To *italicize* text, highlight the text, and then choose the **I** button. (As an alternative, press CTRL+I.) When the **I** button is gray, the text cannot be italicized.

To underline text, highlight the text, and then choose the **U** button. All fonts can be underlined; CTRL+U is the keystroke shortcut.

To overline text, highlight the text, and then choose the **O** button. All fonts can be overlined; CTRL+O is the keystroke shortcut.

UPPER and lower Case

The **Aa** **Uppercase** and **Lowercase** buttons change selected text to all UPPERCASE or all lowercase. There are a couple of keyboard shortcuts that bypass the menu. Select text, and then press the following keys:

CTRL+SHIFT+U converts selected text to all UPPERCASE.

CTRL+SHIFT+L converts selected text to all lowercase.

Font

Select a font name from the Font droplist. The fonts listed are those installed on your computer. The list probably includes all TrueType fonts included with Windows, as well as fonts installed by AutoCAD and other applications. TrueType fonts have a tiny **T** logo in front of them; AutoCAD fonts are prefixed by a tiny Autodesk **A** logo. (Printer fonts, PostScript fonts, and other fonts are not listed, because AutoCAD does not support them directly.)

You can use one or more fonts in one block of mtext, but remember to highlight the text before selecting the name of a font:

- To highlight some text, click and drag it with the cursor.
- To select all text, press **CTRL+A**. Alternatively, right-click the text and choose **Select All** from the shortcut menu.

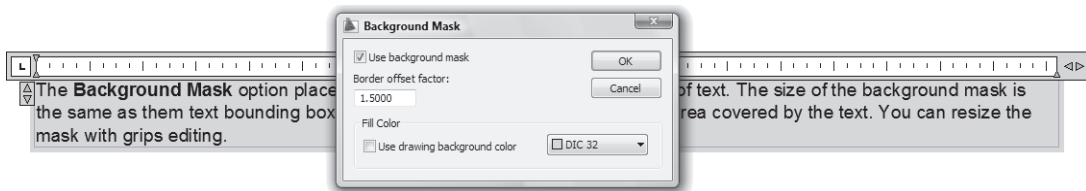
Color

Highlight a portion of the text, and then select a color from the Color list box. You can choose Bylayer, Byblock, the seven basic AutoCAD colors, and the other 16.7 million colors and color books.

Background Mask

The **Background Mask** option places a colored rectangle behind paragraphs of text. The size of the background mask is the same as the mtext window, which may be larger or smaller than the area of the text. You can resize the mask with grips editing.

To create the background mask, select **Background Mask** from the ribbon. In the dialog box, turn on the **Use Background Mask** option, and then select the color.



If you turn on the **Use Drawing Background Color** option, the background color is (usually white), masking out the other drawing entities behind it. You may need to use the **TEXTTOFRONT** command to display text on top of other objects.



Note To create white text on a colored background, select a True Color as follows:

1. In the Text Formatting toolbar's Color droplist, select **Select Color**.
2. In the Select Color dialog box, click the **True Color** tab.
3. Select any color.
4. Drag the **Luminance** slider to the top, turning the text color to white.
5. Click **OK**. The text is white.

Oblique Angle, Tracking, and Width Factor

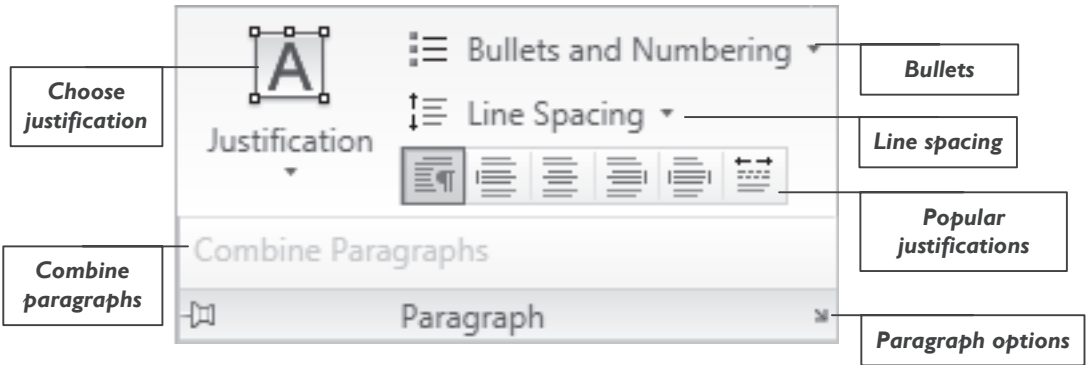
The **Oblique Angle** option changes the slant of selected text, almost like applying italics, except that text can be slanted forward and backwards by a specific amount (up to 85 degrees).

The **Tracking** option changes the distance between characters. By reducing the tracking, you can squeeze more text into a line.

The **Width Factor** option makes selected characters wider and narrower. Again, by making characters narrower, you can squeeze more text into a line. More details follow.

The **STYLE** command illustrates these properties later in this chapter.

Paragraph Panel



Justification

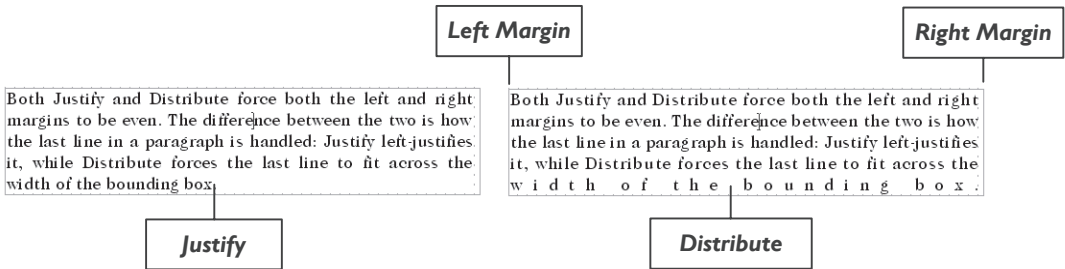
The **Justification** button changes the justification of the entire mtext block relative to the bounding box. You can choose from left, center, and right; and from top, middle, and bottom.

Vertical justification works only when the bounding box is noticeably longer than the paragraphs of text. You would select **Bottom**, for example, when you want the text block to rest at the bottom of the bounding box.

Popular Justifications

The **Horizontal Justification** buttons change the justification of the text relative to the sides of bounding box. You can choose from Left, Center, Right, Justify, and Distribute.

Both **Justify** and **Distribute** force the left and right margins to be even. The difference between the two is how the last line in a paragraph is handled: **Justify** left-justifies it, while **Distribute** forces the last line to fit between the left and right margins.



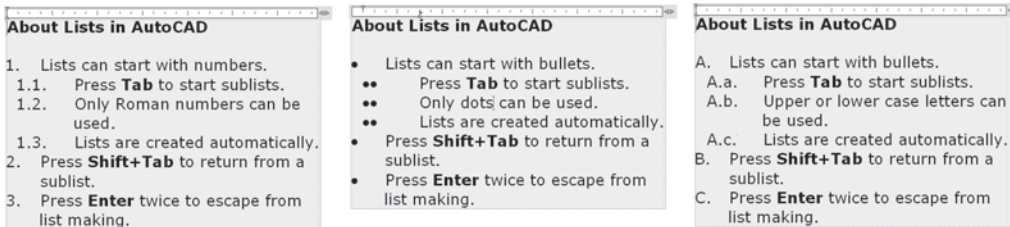
Combine Paragraphs

The **Combine Paragraphs** option groups selected paragraphs into a single paragraph. You need to select at least two paragraphs for this option to operate.

Bullets

The **Bullets** button specifies the format of bullets: letters, numbers, symbols, or none.

AutoCAD automatically numbers (or letters) and indents the text for you. Each paragraph (text that ends with a hard return) is bulleted separately.

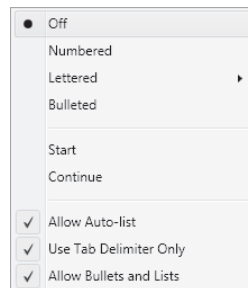


You can use the following characters as punctuation after letter and number bullets: periods (.), commas (,), close parentheses ()), close angle brackets (>), close square brackets (]), and close curly braces (}).

The technical editor advises us how to get this to work:

1. Ensure that all bullet options are turned off.
2. Enter a letter or number, the desired character, and then press **TAB**.
3. Enter the text, and then press **ENTER**. The automatic lettering or numbering turns on.
4. To create sub-bullets, press **TAB** one or more times at the start of the next line.
5. To outdent, press **SHIFT+ TAB**.

Click the **Bullets** button for this list of options:



Off — removes bulleting from the mtext.

Numbered — applies number bullets to paragraphs of text; numbers are Roman numerals only.

Lettered — applies letter bullets to paragraphs of text. Once it reaches Z, it continues with AA, BB, and so on. Letters can be all uppercase or all lowercase.

Bulleted — places round bullets, like this one •.

Start — starts letter bullets from A; starts numbered bullets from 1.

Continue — adds selected paragraphs to the previous list.

Allow Auto-list — applies bullet formatting as you type.

Use Tab Delimiter Only — restricts automatic bullet creation to when the TAB key is pressed.

Allows Bullets and Lists — applies bullets to all text objects that look like lists (namely those that begin with one or more letters or numbers or a symbol, followed by punctuation, a space created by TAB, and end with ENTER or SHIFT+ENTER). When off, this option removes bullet formatting and turns off all other options.

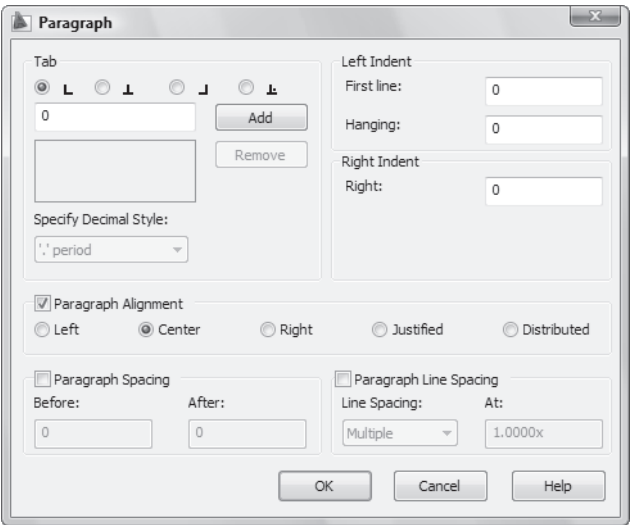
To change the bullet style, click the **Options** button (looks like a downward pointing arrow), and then select **Bullets and Lists** from the menu.

Line Spacing

The **Line Spacing** buttons change the spacing between lines. Select the text, and then choose a spacing. This increases the spacing.

Paragraph Options

The **Paragraph** option (named **Indents and Tabs** in earlier releases of AutoCAD) displays a dialog box that controls all aspects of paragraphs.



The **Tab** section lets you enter exact tab stop positions. *Tab stops* line up columns of data.

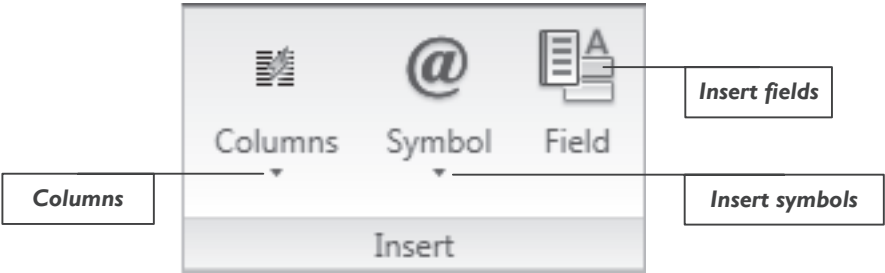
The **Left Indent** and **Right Indent** sections allow you to enter precise distances to indent the first line and the paragraph (i.e. every line).

The **Paragraph Alignment** section selects the type of paragraph justification.

The **Paragraph Spacing** section sets the distance between paragraphs, defined as a distance before and after the paragraph.

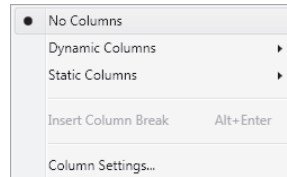
The **Paragraph Line Spacing** section sets the spacing between *lines*, despite the word “paragraph” in the name.

Insert Panel



Columns

The **Columns** button sets the number of columns. It splits the mtext block into two or more columns. This is useful when you need to fit the text into an area that is wider than it is deep.

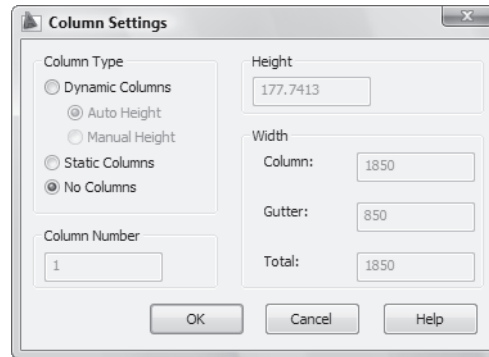


You can choose from the dynamic or static columns, or else force the start of a new column with ALT+ENTER.

Dynamic — AutoCAD creates columns automatically as text is added to the bounding box.

Static — you tell AutoCAD the number of columns you want, 2 or more.

Columns Settings — opens a dialog box that defines all column options in one place.



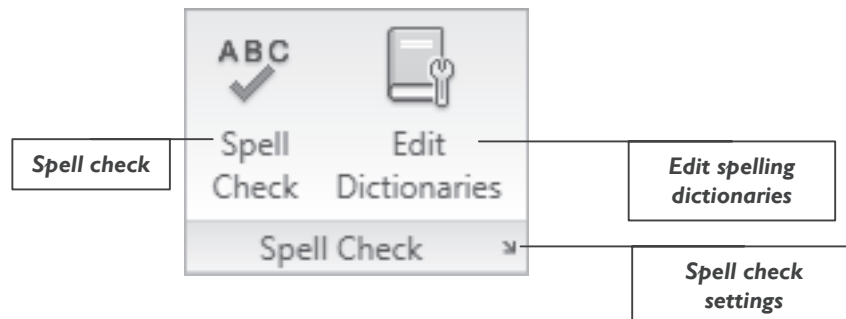
Insert Fields

The **Insert Field** button displays the Field dialog box.

Insert Symbols

The **Insert Symbols** button displays a menu of symbols commonly used in drawings. Click **Other** to display the Windows Character Map dialog box, which provides access to all symbols. More details follow in this chapter. (Ironically, “@” is not one of the available symbols; is available through the keyboard.)

Spell Check Panel



Spell Check

The **Spell Check** button displays the Spell Check dialog box; see the SPELL command later in this chapter.

Tools Panel

Find and Replace

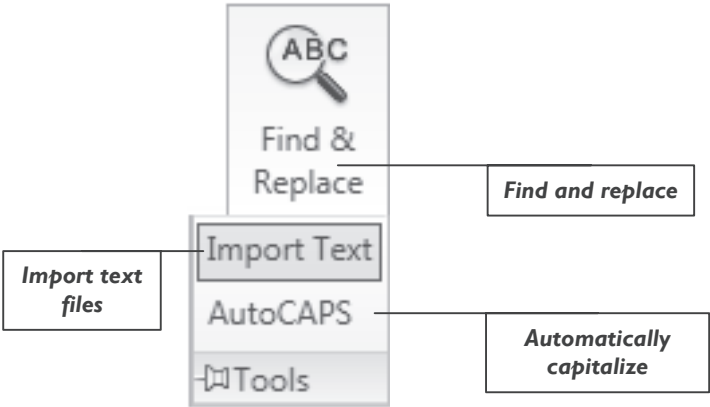
The **Find & Replace** button displays the Find dialog box; see the FIND command later in this chapter.

Import Text

The **Import Text** option displays a dialog box for selecting documents saved in plain text (ASCII format) or RTF (rich text format). The better alternative is to copy text from any document to the Clipboard, and then paste it into the mtext editor. The text formatting is then retained.

The Import Text option cannot import text saved in a word processing format, such as Write (WRI), Word (DOC), or WordPerfect (WP) files. Attempting to import these formats results in a warning.

AutoCAD can import files up to 32KB in size. Excel spreadsheets imported through RTF format are truncated at 72 rows, unless created in Office 2002 (with service pack 2 installed) or later. Text color is set to the current color; some (but not all) other formatting is preserved, such as font names and sizes, though not justification or columns.

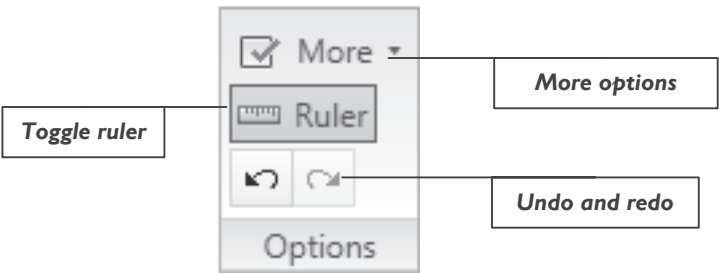


AutoCAPS

The **AutoCAPS** toggle turns on the computer’s CapsLock mode, so that all typed text and imported text are displayed in uppercase characters. If the CapsLock light won’t go off on your keyboard, it’s because this option is still turned on.

(“Or,” suggests the copy editor, “you spilled Gatorade on your keyboard.”)

Options Panel



Toggle Ruler

Click the **Toggle Tab Ruler** button to hide and display the tab ruler, located atop the boundary box.

Undo and Redo

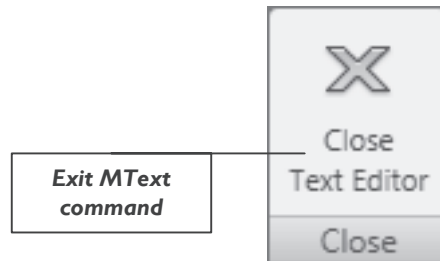
To reverse the last operation, choose the **Undo** button. Or press CTRL+Z. Choose the **Redo** button to “undo” the undo. Or press CTRL+Y.

More

Clicking the **More** button displays a menu of additional options, described later. (This option was formerly known as “Options.”)

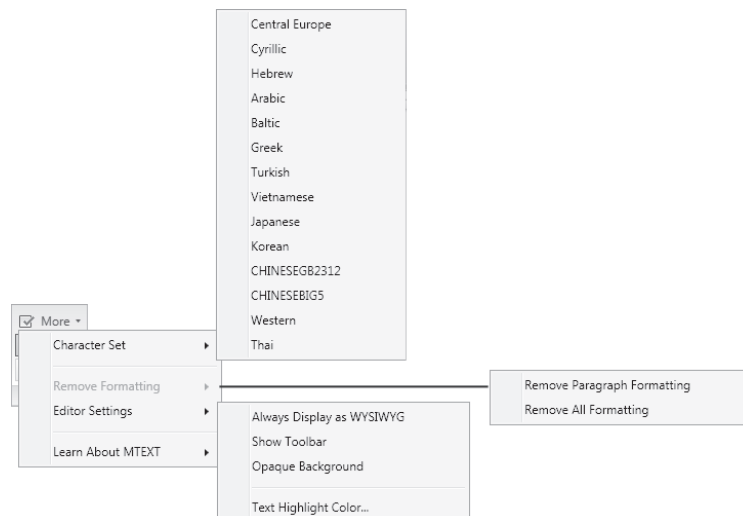
Close Panel

Clicking the **Close Text Edit** button exits the MTEXT command; you can edit the text later by double-clicking it.



MORE OPTIONS

The More button displays the following menus:



Character Set

The **Character Set** option changes the character set to match local language requirements. Not all fonts support all character sets.

Remove Formatting

The **Remove Formatting** option removes bold, italic, and underline formatting from the selected text.

Remove Paragraph Formatting removes formatting from paragraphs, such as justification.

Remove All Formatting removes all format overrides. This option returns the font to the current style.

Editor Settings

The **Editor Settings** item controls the display of elements in the mtext editor.

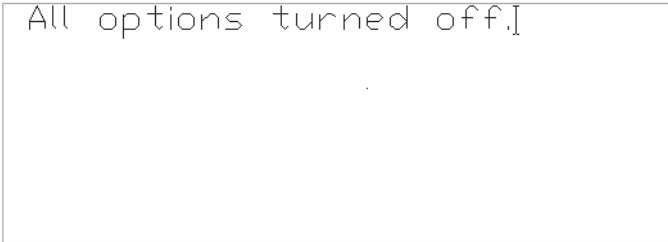
Always Display as WYSIWYG edits text as it is, even when off at an angle.

Show Toolbar toggles the display of the entire Text Formatting toolbar.

Opaque Background — toggles the gray background of the text entry area.

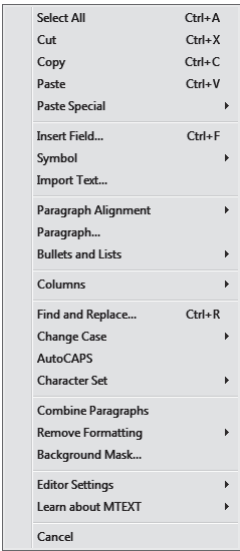
Text Highlight Color — selects a color for highlighted text.

With all options turned off, you have a bare text editor:



MTEXT SHORTCUT MENU

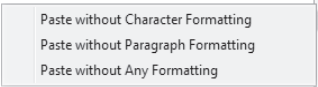
The ribbon does not display all available mtext operations. Right-clicking the text entry area displays a shortcut menu that reveals additional options. The content of the menu varies according to the text selected.



The first four options on the shortcut menu are standard: **Select All** (CTRL+A), **Cut** (CTRL+X), **Copy** (CTRL+C), and **Paste** (CTRL+V).

Paste Special

The **Paste Special** item provides ways to paste text without losing formatting. Its options can be confusing to sort out, not least because Autodesk’s documentation doesn’t explain them.

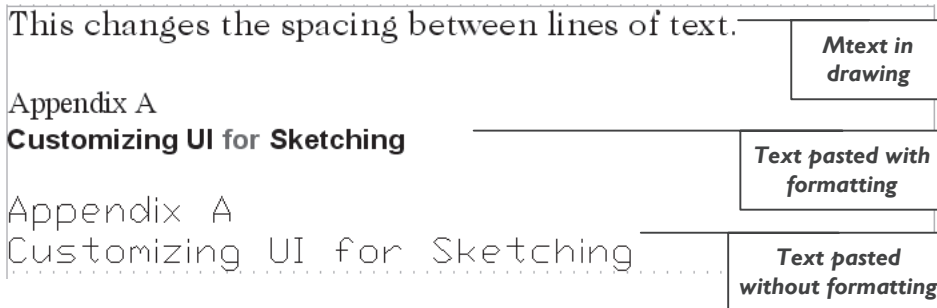


The regular Paste command pastes text with all formatting intact. For instance, if the text contains several fonts and colors, they are preserved when pasted as mtext. Here are the variations on paste:

Paste without Character Formatting pastes the text according to the current AutoCAD text style.

Paste without Paragraph Formatting pastes text without right or centered justification.

Paste without Any Formatting pastes text with the current style.



Insert Field

The **Insert Field** option displays the Field dialog box.

Symbol

The **Symbol** option displays a submenu listing common drafting and other symbols.

Degrees	%%d
Plus/Minus	%%p
Diameter	%%c
Almost Equal	\U+2248
Angle	\U+2220
Boundary Line	\U+E100
Center Line	\U+2104
Delta	\U+0394
Electrical Phase	\U+0278
Flow Line	\U+E101
Identity	\U+2261
Initial Length	\U+E200
Monument Line	\U+E102
Not Equal	\U+2260
Ohm	\U+2126
Omega	\U+03A9
Property Line	\U+214A
Subscript 2	\U+2082
Squared	\U+00B2
Cubed	\U+00B3
Non-breaking Space	Ctrl+Shift+Space
Other...	



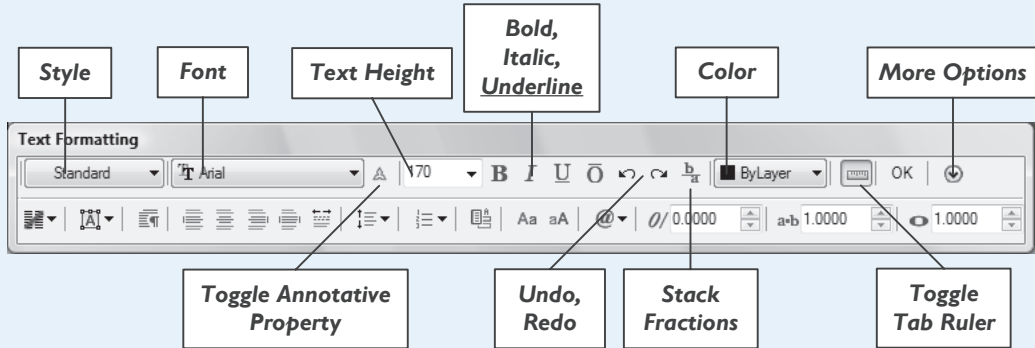
Behind the symbol names are Unicode numbers for each character, such as \U+2248. Some of the symbols can be used for Greek characters; for example, **Ohm** is the same as **omega**.

One symbol is invisible: the “non-breaking space.” When you place it between two words, it prevents AutoCAD from using that space to wrap the sentence. For instance, if your text includes the phrase “one inch,” you might want a non-breaking space between the “one” and “inch” to keep the two words together.

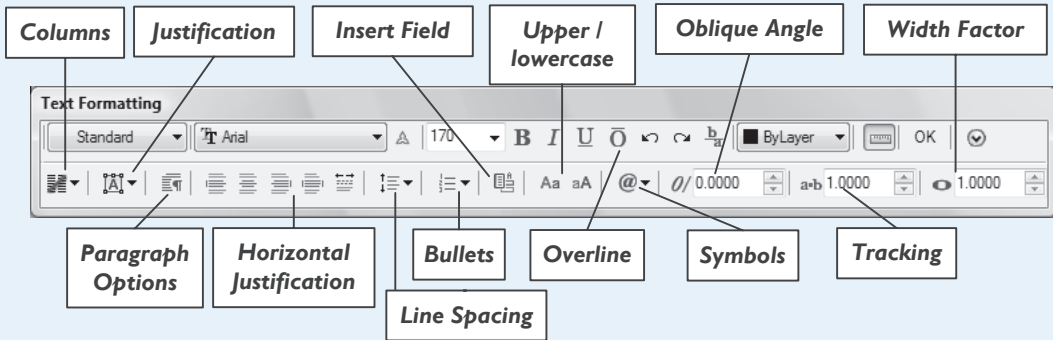
Other opens the Windows Character Map dialog box, which contains all the characters available for the font. Inserting a symbol in this way requires the rather awkward process of (1) selecting the symbol, (2) clicking **Select**, (3) clicking **Copy**, (4) clicking **x** to close the dialog box, (5) right-clicking in the mtext editor, and (6) selecting **Paste**.

TEXT FORMATTING TOOLBAR

The Text Formatting toolbar appears when MTEXTTOOLBAR = 1. The top half of the toolbar provides these functions:



The bottom half of the Text Formatting toolbar shows the following options:



One reason to use the toolbar is that it has the Stack button missing from the ribbon.

Stack

The **b_x** button stacks fractions by changing side-by-side text, like 11/32, to show the 11 over the 32. Using this feature takes these steps:

1. Type the fraction, such as 11/32.
2. Highlight the entire fraction.
3. Choose the **b_x** button.

AutoCAD replaces the numbers with a stacked fraction. Repeat steps 2 and 3 to “unstack” stacked fractions. This button is not limited to stacking numbers: it stacks any combination of text, numbers, and symbols — whatever is in front of the slash goes on top; whatever is behind goes underneath.

Also, you are not limited to working with slash symbols. The carat (^) and the hash or pound mark (#) indicate different kinds of stacking:

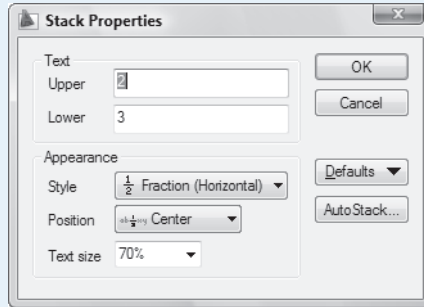
Slash (/) stacks with a horizontal bar.

Carat (^) stacks without a bar (tolerance style).

Pound (#) stacks with a diagonal bar.

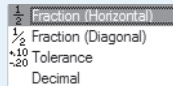


When you right-click stacked text, a new option is added to the shortcut menu: **Stack Properties**. It displays the dialog box illustrated on the next page.

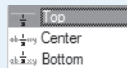


Upper and **Lower** fields specify the fraction text. This lets you change the values.

Style droplist specifies the three slash styles noted above: horizontal, diagonal, and none.

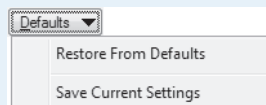


Position droplist specifies the position of the fraction relative to the baseline of text on either side of the fraction: top, center, or bottom.

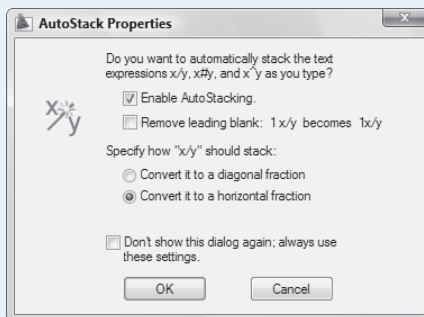


Text size droplist specifies the size of fraction text relative to regular text. The range is from 50% (half as tall) to 100% (same size); the default is 70%.

Defaults button saves the current settings as the default, or restores previously saved defaults.



AutoStack button displays the AutoStack Options dialog box.



Enable AutoStacking option toggles the automatic stacking of numbers on either side of the /, #, and ^ characters.

Remove Leading Blank removes blanks between whole numbers and stacked fractions. This option is available only when AutoStacking is turned on.

Convert It to a Diagonal Fraction creates stacked numbers with the diagonal slash, regardless of the stack character (/, #, and ^).

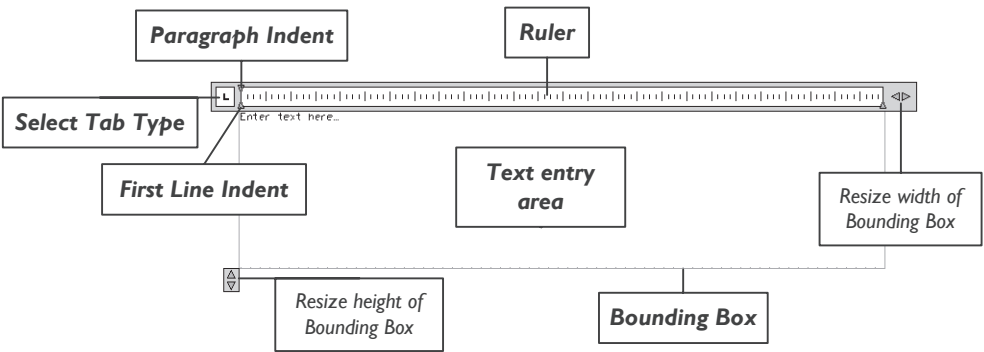
Convert It to a Horizontal Fraction creates stacked numbers with the diagonal slash, regardless of the stack character.



Note When drawings with diagonal stacked fractions are opened in AutoCAD Release 14 or earlier, they are converted to horizontal fractions, but are restored when opened in AutoCAD 2000 or later.

MTEXT TEXT ENTRY AREA

The text entry area is a rectangle that represents the mtext bounding box. The box is topped by a tab bar that sets tabs, indents, and margins. A different indent and tab setting can be applied to each paragraph.



First Line Indent

The *first line indent* shows how far in the first line of each paragraph is indented. You move its arrowhead along the tab bar by dragging it back and forth with the cursor.

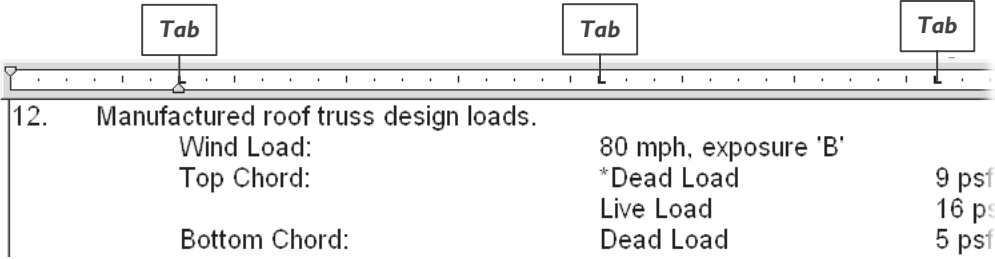
You can also create *hanging indents*: move this indent farther left than the paragraph indent. Hanging indents are created automatically when you apply bullets and lists.

Paragraph Indent

The *paragraph indent* shows how far the entire paragraph should be indented. Only the left indent can be modified; to change the right indent, drag the boundary box margin.

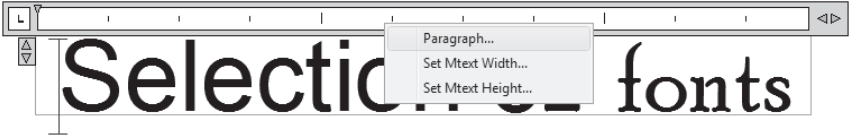
Tabs

Click the tab bar to set tabs; existing tabs can be dragged back and forth along the tab bar. AutoCAD supports the commonly used left tabs (illustrated below), as well as right, center, and tabs for lining up text by decimals.



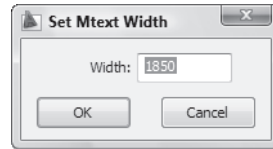
Change the tab type by clicking the **L** icon at the left end of the tab bar.

Right-clicking the tab bar reveals a shortcut menu:



Set Mtext Width and Set Mtext Height

The **Set Mtext Width** and **Set Mtext Height** options display a dialog box that allows you to enter a precise width or height for the bounding box.



As an alternative to this dialog box, you can drag the right edge of the bounding box, making it wider and narrower.

PLACING PARAGRAPH TEXT: ADDITIONAL METHODS

When the MTEXT command starts up, it displays a prompt line with many options. Each time you respond to an option (except the **Width** option), AutoCAD redisplay the following MTEXT prompt — until you pick the opposite corner:

Specify opposite corner or [Height/Justify/Line spacing/Rotation/Style/Width/Columns]:

- **Height** specifies the height of text.
- **Justify** specifies the justification of text in the bounding box.
- **Line spacing** specifies the spacing between lines of text.
- **Rotation** rotates the bounding box.
- **Style** selects the text style.
- **Width** specifies the width of the bounding box.
- **Columns** specifies the number of columns.
- **TEXTTOFRONT** command brings text in front of other objects.

Let's look at each.

Height

The **Height** option changes the height of the text used by the mtext editor. It prompts you:

Specify height <0.2000>: *(Type a number or indicate a height.)*

Justify

The **Justify** option selects the justification and positioning of text within the bounding box. AutoCAD prompts:

Enter justification [TL/TC/TR/ML/MC/MR/BL/BC/BR] <TL>: *(Enter an option, or press ENTER.)*

The justification options are the same as for the MTEXT command, except that in this case, there are two areas where justification applies: (1) text and (2) flow.

Text justification is left, center, or right, relative to the left and right boundaries of the rectangle. *Flow justification* positions the block of text top, middle, and bottom relative to the top and bottom boundaries of the rectangle.

Linespacing

The **Linespacing** option changes the spacing between lines of text, sometimes called the “interline spacing” or “leading.” The option has two methods for specifying the spacing: **At least** and **Exactly**. The **At least** option specifies the minimum distance between lines, while **Exactly** specifies the precise distance between lines of text.

Enter line spacing type [At least/Exactly] <At least>: **a**

Enter line spacing factor or distance <1x>: *(Enter a value, including the x suffix.)*

The value you enter is a multiplier of the standard line spacing distance, which is defined in the font.

Rotation

The **Rotation** option specifies the rotation angle of the bounding box.

Specify rotation angle <default>: *(Enter an angle, or pick two points.)*

This option rotates the entire block of text. For example, enter 90 degrees to place text sideways at the edge of a drawing.



Note When you use the mouse to show the rotation angle, AutoCAD calculates the angle as follows:

- The *start* of the angle is the X-axis.
- The *end* of the angle is the line anchored by the “Specify first corner:” pick; the other corner is now in line with the indicated angle.

Style

The **Style** option selects a text style to use for the multiline text.

Enter style name (or ?) <Standard>: *(Enter a name, or press ? for a list of style names.)*

When you enter ? in response to this prompt, AutoCAD displays the names of text styles defined in the drawing. You create new text styles with the **STYLE** command (described later in this chapter) or with a style borrowed from another drawing using the **DesignCenter**.

Width

The **Width** option specifies the width of the bounding box.

Specify width: *(Enter a value, or pick a point to show the width.)*

A width of 0 (zero) has special meaning. AutoCAD draws the multiline text as one long line (no word wrap), as if there were no bounding box at all.

Columns

The **Columns** option specifies the number of columns.

TEXTTOFRONT

The **TEXTTOFRONT** command brings text to the front of the display order. It works with text and dimensions. (Use the **HATCH** command’s **Draw Order** option for hatch patterns and the **DRAWORDER** command for all other objects.)

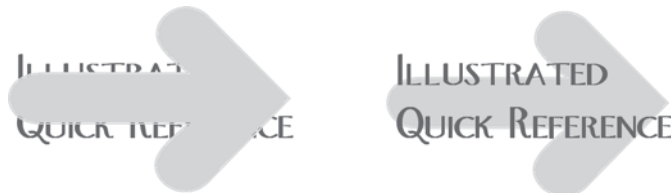
You can force AutoCAD to display all text over top of all other objects in the drawing, like this:

Command: **texttofront**

Bring to front [Text/Dimensions/Both] <Both>: **t**

n object(s) brought to front.

This command is much more efficient than the **DRAWORDER** command, because it selects text only, instead of any object.



Left: Text behind another object.

Right: Text brought to the front.

STYLE

The **STYLE** command defines named text styles based on fonts.

A *style* is a collection of properties applied to the font of your choice, such as slanted text, or perhaps condensed text. Styles are often used in word processing and desktop publishing, and work the same way in AutoCAD. The idea is to preset most text properties so that you don't need to do it each time with every line of text. (Some properties, however, cannot be set by styles, for example color and layer.)

Styles are primarily required by the **TEXT** and **MTEXT** commands, as well as by attributes, fields, leaders, and dimensions. Styles are stored and accessed by names of your choice of up to 256 characters.

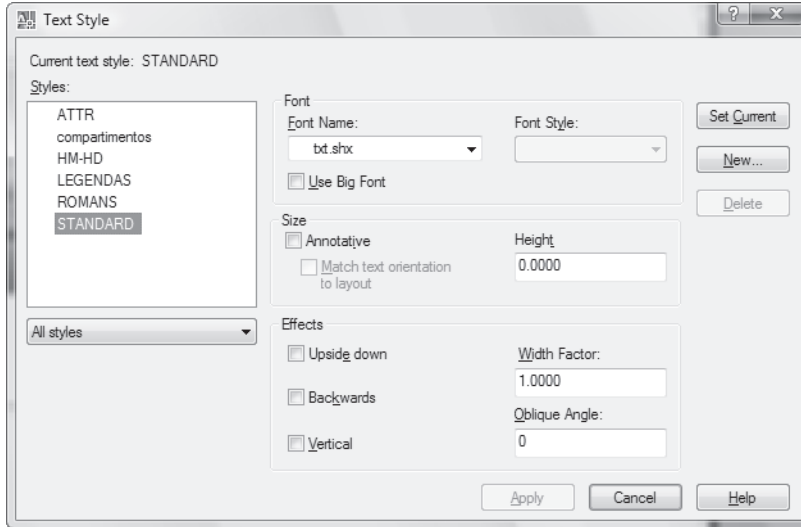
Styles store the following information, and then instantly apply them to text. (Not all fonts support all properties.)

- **Font file** associates a *.ttf* (TrueType) or *.shx* (AutoCAD) file with the style. PostScript *.pfb* font files can be used after conversion to *.shx* with the **COMPILE** command.
- **Font style** selects regular, boldface, or italicized text.
- **Height** specifies the height of the text. Normally, this is set to 0, which means that the **TEXT** command prompts you for the height. Entering a height here means the command doesn't pester you later.
- **Annotative** toggles annotative scaling.
- **Orientation** makes text independent of drawing and layout rotation.
- **Width factor** specifies a multiplier making the text wider or narrower. A width of 1 is standard; a width factor of 0.5 produces text at one half the width.
- **Obliquing angle** determines the slant of the text. A positive angle produces a forward slant; a negative angle, a backward slant.
- **Backwards** mirrors the text.
- **Upside down** places the text upside down.
- **Vertical** places the text vertically.

When you change a style, all text assigned to that style also changes.

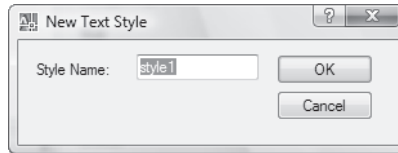
TUTORIAL: DEFINING TEXT STYLES

1. To define one or more text styles, start the **STYLE** command:
 - In the ribbon's Annotate tab, choose **Manage Text Styles** in the Text panel's style name droplist.
 - At the 'Command:' prompt, enter the **STYLE** command.
Command: **style** (Press ENTER.)
 - Alternatively, enter the aliases **st** or **ddstyle** (the old name) at the 'Command:' prompt.
2. In all cases, AutoCAD displays the dialog box:



The **Styles** list provides the names of styles already in the drawing. Use this list to change the properties of existing styles. (Options that do not apply to a font or style are grayed out.)

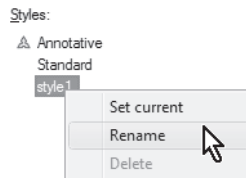
3. To create a new style, click **New**. Notice that AutoCAD displays the New Text Style dialog box. This is where you name the style.



Change the default name “style1” to anything else — up to 255 characters long — and then choose **OK**. Notice that AutoCAD adds the name to the **Styles** list.



Notes To rename a style, right-click its name, and then choose **Rename** from the shortcut menu. You can rename all styles except Standard.



The **Delete** option erases selected styles from drawings — but only if they are unused. If a style is used by any amount of text, AutoCAD complains, “Style is in use, can’t be deleted.” The Standard style can never be deleted.

4. From the Font Name droplist, select a font.
There are many font files available for AutoCAD, which comes with an large selection: specialty fonts for map making (symbols instead of letters), cursive writing, and so on.

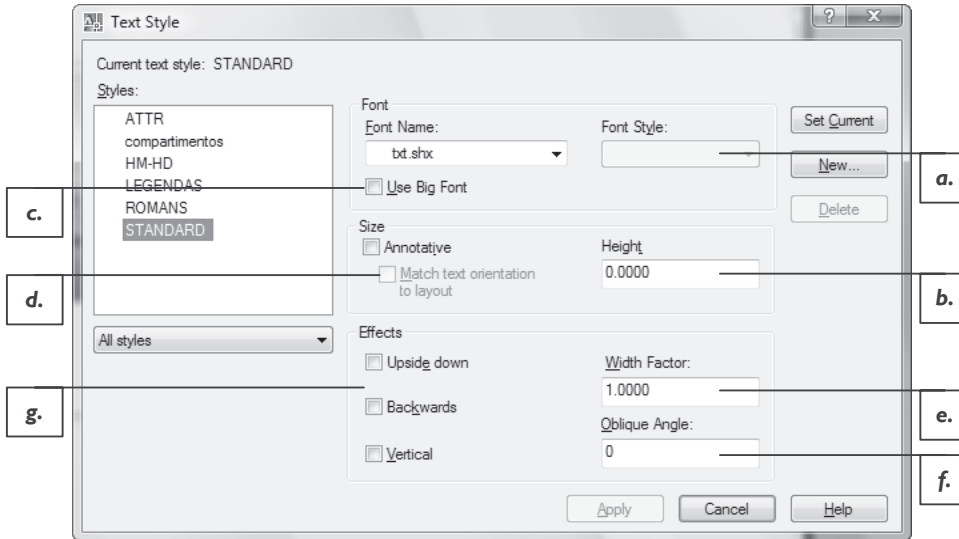


Note The font file should be appropriate to the application. Most mechanical applications use the ANSI type lettering, often called “Leroy.” The Leroy lettering machines were made by K&E, and permitted precise lettering on hand-drawn plans. The Simplex font (also known as “RomanS” — roman, single stroke) closely approximates this style.

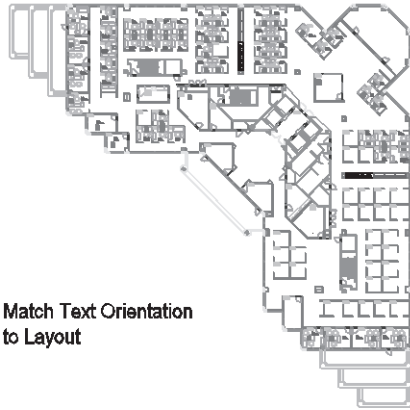
Architectural drawings typically use “hand-lettered” fonts, such as City Blueprint. Engineering applications often use the Simplex font, since it closely resembles a traditional font constructed with a lettering template.

Multistroke fonts use a code, with “S” denoting single stroke, “D” duplex or double stroke, “C” complex (also double stroke), and “T” triple stroke.

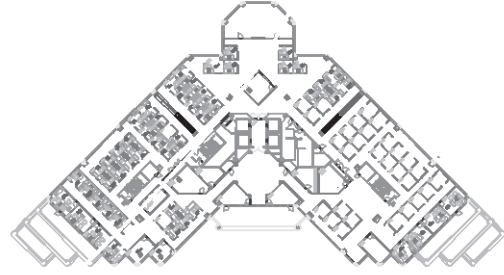
5. Select any of the following options, if available (not grayed out):



- a. The Font Style droplist selects Regular, **Boldface**, *Italic*, or **Bold Italic**. Not all font styles are available, because some fonts cannot handle these effects.
- b. The **Height** text box specifies the height of the text, which is measured from the baseline to the top of uppercase letters. There are three ways to specify the height:
 - Determine the scale of the drawing before deciding the text height, as described earlier in this chapter.
 - Enter a value of 0. This gives you the flexibility to change the height as the text is being placed. This applies particularly if you are not sure of the scale until the end of the drawing process.
 Turn on the **Annotative** option, and then specify the true (paper) height.
- c. The **Use Big Font** option refers to a class of AutoCAD fonts that handle more than 256 characters, primarily for Chinese and other languages with thousands of characters.
- d. **Match Text Orientation to Layout** ensures the text is always displayed “correctly,” no matter how the rest of the drawing is rotated. The name of this option is somewhat misleading, because it works even when the layout is not rotated. In the figure below, the entire drawing (including the text) is rotated 90 degrees. The text keeps its orientation, which makes it easier to read.



Match Text Orientation to Layout



Match Text Orientation to Layout

- e. The **Width Factor** determines the width of characters. A width factor of 1.0 is “standard.” A decimal value, such as 0.85, draws text narrower by 15%, which is useful for condensed text. Values greater than 1 expand text.

0.5 Width Factor

2.0 Width Factor

- f. **Oblique Angle** determines the slant of characters. A zero obliquing angle draws text that is “straight up.” Positive angles, such as 15 degrees, draw text that slants forward, while negative angles draw text that slants backward. Valid values are between –85 and 85 degrees.

15° Oblique

-15° Oblique

- g. **Upside Down** draws text upside down.

Upside Down Text

Backwards means the text is drawn in reverse. Backwards text is useful for plotting drawings on the back side of clear media, or for mold drawings used by the casting and injection-molding industries.

Reverse Text

Vertical draws text vertically, which is *not* the same as rotating text by 90 degrees. Text is drawn so that each letter is vertical and the text string itself is vertical. TrueType and some AutoCAD fonts cannot be drawn vertically.

Vertical



Note Obliquing, underscoring, or overscoring should not be used with vertical text, since the result will look incorrect.

6. Click **Set Current** to make the style current; this means any text you draw now takes on this style.
7. Click **OK** to exit the dialog box.

USING TEXT STYLES

The difference between fonts and styles is sometimes confusing. You cannot use fonts directly in AutoCAD; fonts can only be used through styles. (The exception is in the mtext editor, but even there the font only overrides a style.) Styles modify the properties of a font, such as its height, width, and slant. One font can be used by many styles, but each style can specify only one font.

1. To use a font, you must create a style with the **STYLE** command.
2. Then, in the **TEXT** and **MTEXT** commands, you specify the style to use for the text you place.

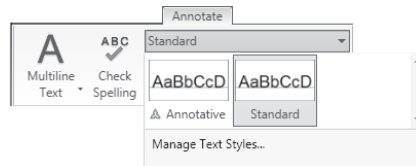
Each line or paragraph of text is assigned a style; you may change the style with the **PROPERTIES** command. When the style changes, the look of the text also changes. To change text globally, change the style with the **STYLE** command.

Often, drafting offices create a standard selection of styles, each of which is used for a particular type of text in drawing. For example, one style is used for notes, another for titles, and others for the different parts of the title block. Using styles ensures consistency across drawings, no matter the drafter.

Styles Panel

An alternative method to applying styles is using the ribbon's Home | Annotation panel.

1. Select one or more lines of text.
2. Pick a style name from the style droplist:



Notice that AutoCAD changes the look of the text to match the style. In the panel, clicking the **Text Style** button displays the Text Style dialog box.

DEFINING STYLES: ADDITIONAL METHODS

Several system variables affect the look and quality of TrueType fonts.

- **FONALT** specifies font substitution.
- **TEXTFILL** toggles the fill of TrueType fonts for plots.
- **TEXTQLTY** adjusts the quality of TrueType fonts for plots.

Let's look at each.

FontAlt

The **FONALT** system variable specifies the name of the font to use when other fonts cannot be found.

Often, different computers have different collections of installed fonts. When you receive a drawing from another AutoCAD drafter, the drawing may use fonts not found on your computer. So that text doesn't go missing, AutoCAD substitutes the *simplex.shx* font for missing fonts. (If the Simplex font cannot be found, AutoCAD displays the Alternate Font dialog box, so that you can select another font file.)

Command: **fontalt**

Enter new value for FONALT, or . for none <"simplex">: (Enter the name of a TrueType or AutoCAD font.)

For missing TrueType fonts, AutoCAD can substitute another font without warning you, which may be annoying. On my computer, AutoCAD substitutes the Absalom font, which makes drawing texts

hard to read, like this:

New paragraph with
unstacked 11/32 and stacked
13
fractions 64.

The technical editor notes that problem fonts can be caused by the setting of AutoCAD's FONTALT system variable, the contents of the *acad.fmp* file, and/or font substitutions made by Windows.

TextFill

The TEXTFILL system variable toggles the fill of TrueType fonts for plots. When on (the default), fonts are filled; when off, only the outline of the font is plotted. This system variable affects only the plotting of fonts; it does not affect the display of fonts in drawings.

Command: **textfill**

Enter new value for TEXTFILL <1>: **0**

Outline Font

TextQlty

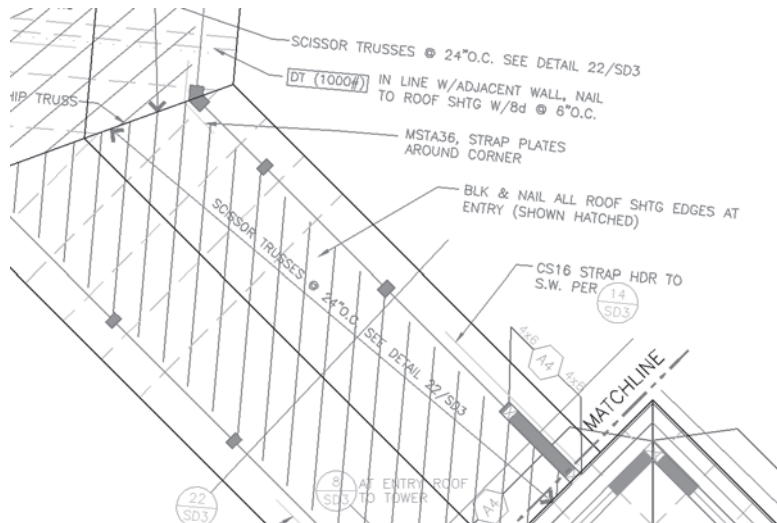
The TEXTQLTY system variable specifies the resolution of TrueType font outlines (technically, the tessellation fineness) when plotted: 0 means the text is not smoothed; 100 means maximum smoothness. The difference is seen only when drawings are plotted; this system variable does not affect the display.

Command: **textqlty**

Enter new value for TEXTQLTYL <50>: **100**



The QLEADER command places *callouts* (a.k.a leaders) in drawings. The figure below illustrates examples of leaders in a drawing.



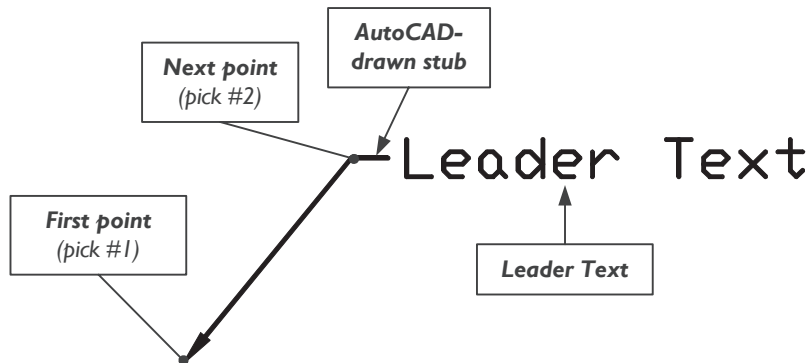
AutoCAD has several commands for creating leaders. The differences among them are as follows:

- **QLEADER** (short for “quick leader”) — includes a Settings dialog box for fashioning the leader’s properties. This is the command you’ll want to use most of the time.
- **MLEADER** (short for “multiline leader”) — groups multiple leader lines and block annotation together. Mleaders can have preset styles created through the **MLEADERSTYLE** command, and can collect multiple leaders into one. Unlike leaders and qleaders, mleaders are limited to mtext and blocks; they do not include tolerances or selected objects.
- **LEADER** — all options are entered at the command line. This is an earlier method of creating leaders, and has been left in AutoCAD for compatibility reasons.
- **DIM: LEADER** — options are also entered at the command line, and is the original method.

TUTORIAL: PLACING LEADERS

In the following tutorial, you draw a rectangle, and then specify its size through a callout, placed as a leader.

1. Open a new drawing in AutoCAD with the *acad.dwt* template.
2. Use the **RECTANGLE** command to draw a rectangle of any size.
3. To place the callout in the drawing, start the **QLEADER** command:
 - At the ‘Command:’ prompt, enter the **QLEADER** command:
Command: **qleader** (Press ENTER.)
 - Alternatively, enter the **le** alias at the ‘Command:’ prompt.
4. AutoCAD prompts you for the *first point*, which is the tip of the leader’s arrowhead:
Specify first leader point, or [Settings] <Settings>: (Pick a point near the rectangle, shown as point 1 in the following figure.)



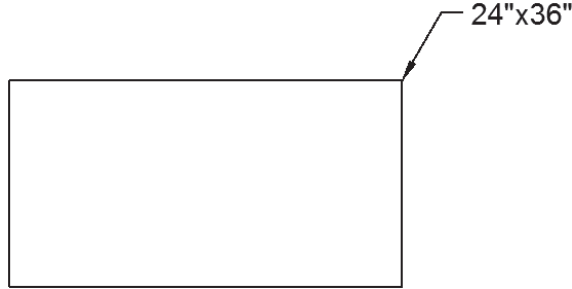
5. AutoCAD prompts you to pick the *next point*.
Specify next point: (Pick point 2.)
From now on, each of your picks indicates the location of the next vertex in the leader line — until you pressing **ENTER** to stop drawing it. (Later, AutoCAD will add the short horizontal stub automatically.)
Specify next point: (Press **ENTER** to end drawing the leader line.)
6. Enter 0 for unconstrained width. (If the width of the text must be constrained, enter the width here.)
Specify text width <0.0000>: (Press **ENTER**.)

7. Enter the text, and then press **ENTER**. For this tutorial, enter the size of the rectangle, such as 24"x36".

Enter first line of annotation text <Mtext>: (Enter text, such as 24"x36".)

8. Press **ENTER** twice in a row to exit the command.

Enter next line of annotation text: (Press **ENTER** to exit command.)



If the leader's size overwhelms the drawing, use the **DIMSCALE** system variable to change the size prior to placing the leader. To make the leader text and arrowhead smaller, use values between 0 and 1; for larger leaders, use values above 1.

If you use object snap to touch the leader's start point to an object, the leader is *associated* with the object. This means that when you move the object, the arrowhead moves with the object, although the leader text stays in place. (Associativity is available when associative dimensioning is turned on through the **DIMASSOC** system variable — which is the case in most drawings.)

Many of the leader's properties, such as line color and arrowhead scale, are determined by system variables related to dimensioning, all of which start with **DIM**. To modify the properties, use the **DIMSTYLE** command, as described in the next chapter.

You can use grips editing to modify the leader's position. To edit the leader text, double-click the text (not the leader line), and AutoCAD displays the familiar mtext editor.

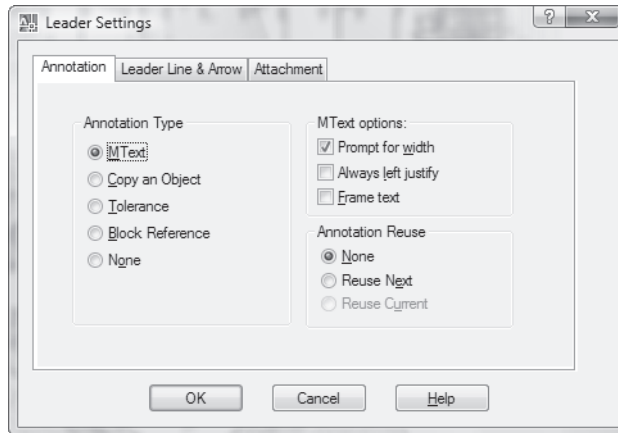
PLACING LEADERS: ADDITIONAL METHODS

The **Settings** option displays a dialog box for specifying the look and operation of “q”-leaders.

Command: **qlleader** (Press **ENTER**.)

Specify first leader point, or [Settings] <Settings>: (Type **S**.)

Notice the dialog box:



Annotation Type

The options you select under **Annotation Type** affect the prompt displayed by the **QLEADER** command.

MText (default) — displays prompts for creating leader text from mtext:

Specify text width <0.0000>: *(Enter a width, or press ENTER.)*

Enter first line of annotation text <Mtext>: *(Enter text.)*

Enter next line of annotation text: *(Press ENTER to end the command.)*

Copy an Object — prompts you to select another line of text already in the drawing. This can be multiline text (including other leader text), single-line text, a tolerance object, or a block.

Select an object to copy: *(Select one object.)*

Tolerance — displays the Tolerance dialog box when the command requires annotation; see Chapter 16, “Geometric Dimensioning and Tolerancing.”

Block Reference — prompts you to insert a block in place of the leader text:

Enter block name or [?]: *(Enter the name of a block)*

Specify insertion point or [Scale/X/Y/Z/Rotate/PScale/PX/PY/PZ/PROtate]: *(Pick a point.)*

Enter X scale factor, specify opposite corner, or [Corner/XYZ] <1>: *(Enter a scale factor.)*

Enter Y scale factor <use X scale factor>: *(Press ENTER.)*

Specify rotation angle <0>: *(Enter an angle.)*

None — removes extra prompts; draws the standard arrowhead and leader line.

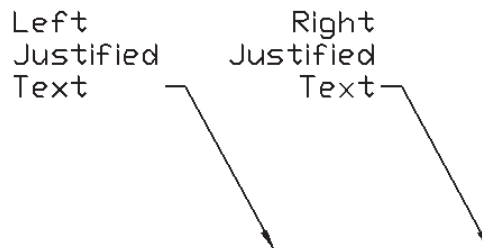
MText Options

The **MText Options** are available only when the **MText** option was selected (see above).

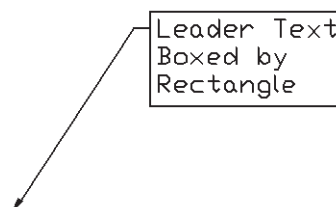
Prompt for Width (default = on) — toggles the display of prompts for the width of the mtext leader text:

Specify text width <0.0000>: *(Enter a width, or press ENTER.)*

Always Left Justify (default = off) — toggles left-justification of mtext, regardless of leader location. AutoCAD normally adjusts the text justification based on the leader line's orientation. When this option is turned on, multiple lines of text are left-justified when the leader angles left, as illustrated by the figure below. When this option is turned off, multiple lines of text are right-justified.



Frame Text (default = off) — toggles the addition of a rectangular frame around the mtext annotation:



Annotation Reuse

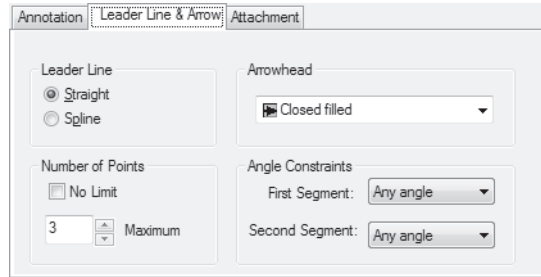
The options under **Annotation Reuse** determine whether the leader text or other annotation is reused by subsequent QLEADER commands.

None (default) — causes AutoCAD to prompt you for an annotation, as described above.

Reuse Next — uses the next annotation you create for subsequent leaders. There is no prompt for specifying mtext, blocks, tolerances, or copying objects.

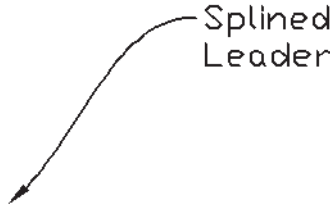
Reuse Current — reuses the current annotation for future leaders.

Leader Line



Straight — draws the leader line using straight line segments.

Spline — draws the leader from a spline object, resulting in a curved leader.



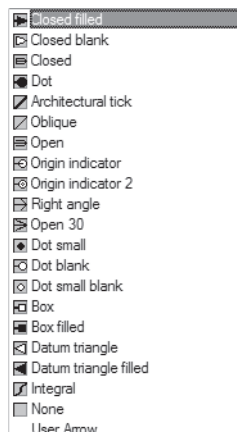
Number of Points

No Limit — command prompts you to “Specify next point:” until you get tired and press ENTER.

Maximum — command stops prompting you after a fixed number, such as 3. You can always stop earlier by pressing ENTER. You must set the number to one more than the number of leader segments you want to create: 3 means that you are prompted for two segments; recall that AutoCAD draws the third segment automatically. The minimum maximum is 2; maximum is 999!

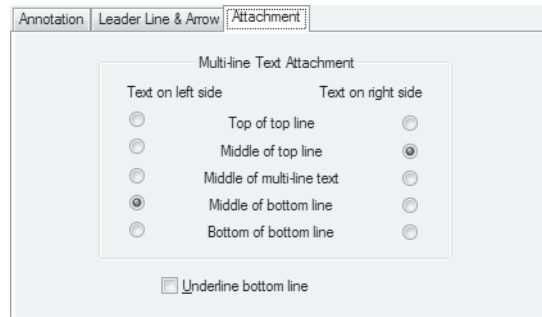
Arrowhead

Select one of the arrowheads shown by the droplist:



User Arrow — lists blocks in the drawing, one of which can be selected for use as the leader's arrowhead.

Angle Constraints

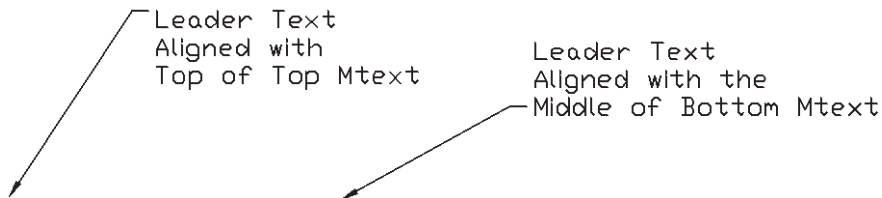


First Segment — specifies the angle of constraint for the first segment. “Any angle” means the segment is not constrained; otherwise, select from Horizontal (0 degrees), 90, 45, 30, or 15 degrees.

Second Segment — specifies the angle-of-constraint for the second leader segment.

Multiline Text Attachment

The **Multiline Text Attachment** options are available only when the **Mtext** option is selected on the **Annotation** tab. These options align the text with the leader line, and can be set differently for left- and right-justified text.



Top of Top Line — aligns the leader line with the top of the top mtext line.

Middle of Top Line — aligns the leader line with the middle of the top mtext line.

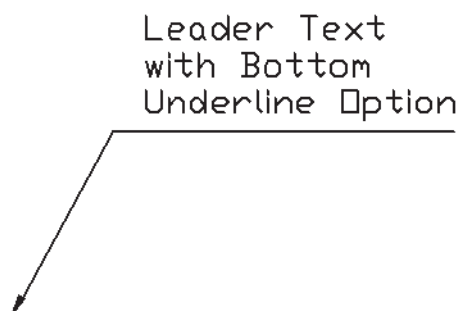
Middle of Multiline Text — aligns the leader line with the middle of the mtext.

Middle of Bottom Line — aligns the leader line with the middle of the bottom mtext line.

Bottom of Bottom Line — aligns the leader line with the bottom of the bottom mtext line.

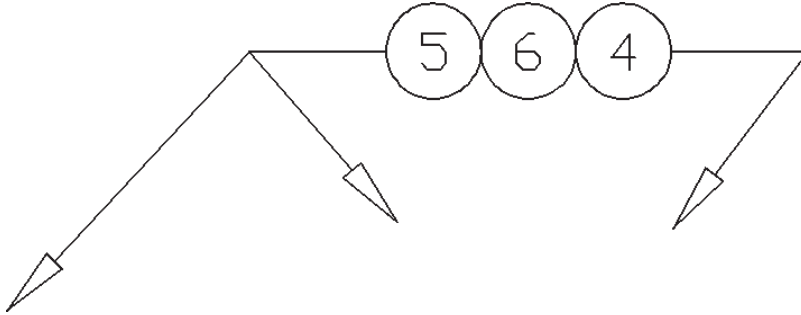
If none of the above options works for you, you may have selected the wrong side: select an option for the correct side.

Underline Bottom Line — attaches the leader line to the bottom of the mtext, and underlines the last mtext line.



MLEADER

The **MLEADER** command draws leaders with multiple lines, among other effects, such as the ones illustrated below.



The initial mleader can be drawn just like a qleader, and then you can add more leader lines, combine annotation from multiple leaders into a single leader, and so on. The **MLEADEREDIT** command adds and removes leader lines; the **MLEADERCOLLECT** command collects annotations. The **MLEADERSTYLE** command determines the default look of mleaders.

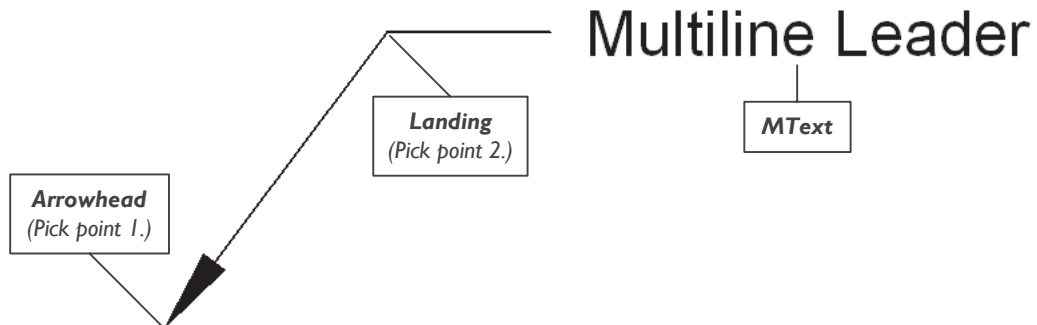
Multiline leaders can have text or blocks, but not tolerance symbols. By default, the blocks are the bubbles illustrated above, but can also be any user-defined block. In addition, mleaders can have multiple blocks per leader line.

When the drawing contains a number of mleaders near each other, the **MLEADERALIGN** command tidies them up by lining them up. Mleaders support annotative scaling, so that they appear at the correct size in model space.

TUTORIAL: DRAWING MULTILINE LEADERS WITH TEXT (MLEADER)

1. To draw leaders with multiple leader lines, start the **MLEADER** command:
 - In the ribbon's Home tab, choose the **Multileader** button in the Annotate panel. Or in the Multileaders panel of the Annotative tab.
 - At the 'Command:' prompt, enter the **multileader** command:
Command: **mleader** (Press ENTER.)
 - Alternatively, enter the **mld** alias at the 'Command:' prompt.
2. In all cases, AutoCAD prompts you to specify the start point of the leader:

Specify leader arrowhead location or [leader Landing first/Content first/Options] <Options>:
(Pick point 1.)



3. The prompts that follow vary, depending on the mleader style. The default style, “Standard,” next prompts you to pick the end of the leader line, which also happens to be the start of the landing line:
Specify leader landing location: *(Pick point 2.)*
4. As soon as you pick the landing location, the mtext editor opens — either in the floating toolbar or on the ribbon, depending on the setting of the **MTEXTTOOLBAR** system variable (as described earlier in this chapter).



Enter text for the leader, and then click the mtext editor’s **OK** button to exit the command.

MLeader Command Options

This command has the following options:

Specify leader arrowhead location or [leader Landing first/Content first/Options]
<Options>:

Specify leader arrowhead location — specifies the endpoint of the arrowhead; the rest of the leader is drawn from there.

leader Landing first — draws the landing first, followed by the arrowhead location.

Content first — draws the “content” (mtext or blocks) first, followed by the arrowhead location.

Options — leads you to a further set of options. Quite frankly, it is easier to set these options with the **MLEADERSTYLE** command, but for the sake of completeness I include them here:

Enter an option [Leader type/leader lAnding/Content type/Maxpoints/First angle/Second angle/eXit options] <eXit options>:

Leader type — draws the leader from straight lines, a spline, or draws none at all.

leader lAnding — specifies the length of the landing, or no landing at all.

Content type — specifies mtext, a block, or no content. If a block, then specifies its name. After the **MLEADER** command has been used at least once in a drawing, then these bubble blocks become available:

_DetailCallout	_TagBox	_TagCircle
_TagSlot	_TagTriangle	_TagHexagon

Maxpoints — specifies the maximum number of leader vertices; default and minimum = 2.

First angle — constrains the angle of the first leader segment.

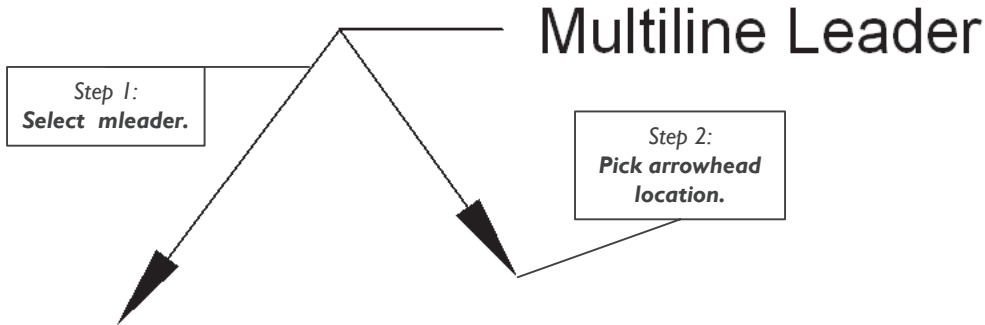
Second angle — constrains the angle of the second leader segment; default = 0 degrees (horizontal).

TUTORIAL: ADDING LEADERS TO MLEADERS (MLEADEREDIT)

Once one multiline leader is in a drawing, you can add additional leader lines to it with the **MLEADEREDIT** command. This command adds and removes leader lines; it works with one mleader object at a time.

1. To add leaders to mleaders, start the **MLEADEREDIT** command:
 - In the ribbon’s Home tab, choose the **Add Leader** or **Remove Leader** button from the Annotation panel.

- At the 'Command:' prompt, enter the **MULTILEADEREDIT** command:
Command: **mleaderedit** (Press **ENTER**.)
 - Alternatively, enter the **mle** alias at the 'Command:' prompt.
2. In all cases, AutoCAD prompts you to select an mleader:
Select a multileader: (Pick an mleader.)
 3. Specify whether you want to add or remove leader lines; for this tutorial, press **ENTER** to take the default, Add:
Select an option [Add leader/Remove leader] <Add leader>: (Press **ENTER**.)
 4. Indicate where you want the arrowhead located; AutoCAD then positions the leader line:
Specify leader arrowhead location: (Pick a point.)



Note The second leader line can end up on the other side of the text, depending on how you position the cursor.

5. Add additional leader lines, or press **ENTER** to exit the command:
Specify leader arrowhead location: (Press **ENTER**.)

To remove leader lines, restart this command and then select the **Remove** option. Pick the leaders to be removed, as follows:

Command: **mleaderedit**
 Select a multileader: (Pick a multileader.)
 Select an option [Add leader/Remove leader] <Add leader>: **r**
 Specify leaders to remove: (Pick one or more leader lines.)
 Specify leaders to remove: (Press **ENTER** to exit the command.)



Notes The **Remove** option allows you to remove all leaders from mleaders, leaving just the mtext or block. Since it is still seen as an mleader object, you can add leader lines to it again.

The Multileader toolbar and panel don't use the **MLEADEREDIT** command to add and remove leaders. Instead, they use a pair of commands undocumented by Autodesk:

AIMLEADEREDITADD — adds leaders to mleaders.
AIMLEADEREDITREMOVE — removes leaders from mleaders.

The advantage of this pair of commands is that they work more quickly, because they exclude the "Select an option [Add leader/Remove leader]" prompt.

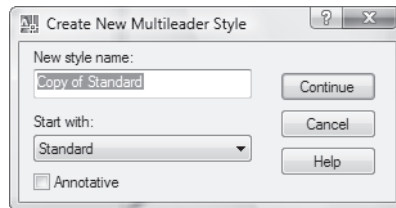
TUTORIAL: DRAWING MLEADERS WITH BUBBLES (MLEADERSTYLE)

Multiline leaders can show bubbles in place of text. To do so, you first need to set up a new mleader style. This is done with the **MLEADERSTYLE** command.

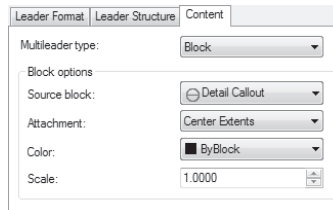
1. To create a new mleader style, start the **MLEADERSTYLE** command:
 - In the ribbon's Annotate tab, choose the **Multileader Style** arrow in the Multileader panel.
 - At the 'Command:' prompt, enter the **mleaderstyle** command:

Command: **mleaderstyle** (Press ENTER.)

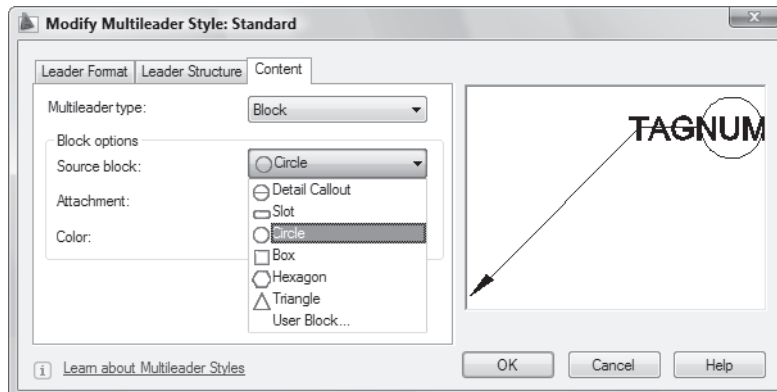
- Alternatively, enter the **mls** alias at the 'Command:' prompt.
2. In all cases, AutoCAD displays the Multiline Leader Style dialog box. To create the new style, click **New**.
 3. Enter a name for the style. For this tutorial, enter "Round Bubble," and then click **Continue**.



4. Since we want to change the content of the mleader, choose the **Content** tab.
5. From the Multileader Type droplist, select **Block**.



6. Notice that the dialog box changes to show options related to blocks. From the Source Block droplist, select **Circle**.



7. Click **OK** to exit the first dialog box.
8. Back in the original dialog box, click **Set Current** to make this new style current.

9. Click **Close** to exit the dialog box.

10. Use the **MLEADER** command to place multiline leaders with the new style.

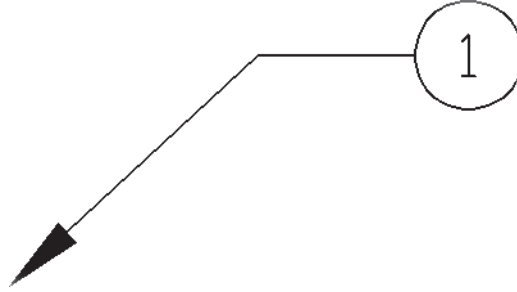
Command: **mleader**

Specify leader arrowhead location or [leader Landing first/Content first/Options]
<Options>: *(Pick a point.)*

Specify leader landing location: *(Pick another point.)*

Enter attribute values

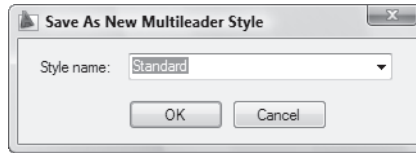
Enter tag number <TAGNUMBER>: **1**



Notes The other style options are very similar to those of the Settings dialog box found in the **QLEADER** command. The **CMLEADERSTYLE** system variable stores the name of the current mleader style name.

You can easily create styles from existing mleaders that you've modified:

1. Select the mleader, and then right click.
2. From the shortcut menu, select **Multileader Style**, and then **Save As New Multileader Style**.
3. Notice the dialog box. Enter a name, and then click **OK**.



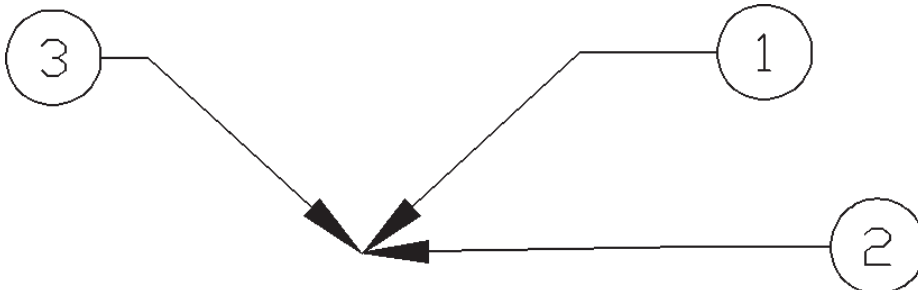
To use the new style, select it from the droplist in the Multiline toolbar or panel. Styles created this way remember the mtext, and will prompt you:

Overwrite default text [Yes/No] <No>: *(Type Y or N.)*

TUTORIAL: COLLECTING BUBBLES INTO ONE MLEADER (MLEADERCOLLECT)

Multiline leaders can have two or more bubbles each. To have multiple bubbles, you first create a number of mleaders with a single bubble each, and then use the **MLEADERCOLLECT** command to combine them. This is commonly done when the leaders all point to the same feature, as illustrated below.

1. Draw two or more bubble mleaders following the steps described above.

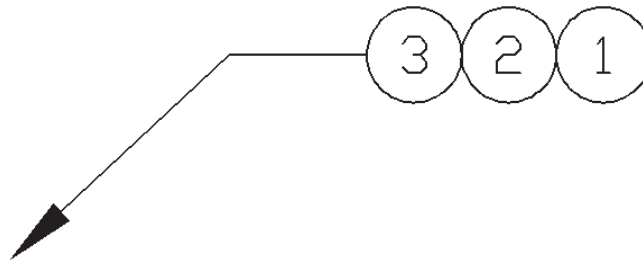


2. To collect multiple multiline leaders into one, start the **MLEADERCOLLECT** command:
 - From the Multileader toolbar, choose the **Collect Multileaders** button.
 - In the ribbon's Annotate tab, choose the **Collect** button in the Multileader panel.
 - At the 'Command:' prompt, enter the **MLEADERCOLLECT** command:
 Command: **mleadercollect** (Press ENTER.)
 - Alternatively, enter the **mlc** alias at the 'Command:' prompt.
3. In all cases, AutoCAD prompts you to select two or more mleaders:

Select multileaders: **all**

Select multileaders: (Press ENTER to exit object selection.)
4. AutoCAD asks where you would like to position the collected mleader:

Specify collected multileader location or [Vertical/Horizontal/Wrap] <Horizontal>: (Pick a point, or enter an option.)



MLeaderCollect Options

The MLEADERCOLLECT's options include:

Specify collected multileader location or [Vertical/Horizontal/Wrap] <Horizontal>:

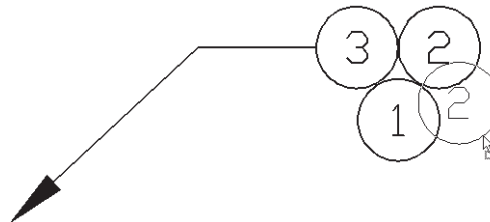
Vertical — stacks the bubbles vertically.

Horizontal — aligns the bubbles horizontally, as illustrated above.

Wrap — wraps the bubbles; AutoCAD asks you how many or how far to wrap them:

Specify wrap width or [Number]: (Enter a distance, or the number of bubbles.)

You can always do the stacking or wrapping yourself, because the added bubbles (#2 and #1, in the figure above) are not part of the leader with #3.

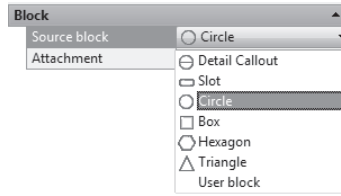


You can drag the bubbles to new locations, as required (illustrated above).

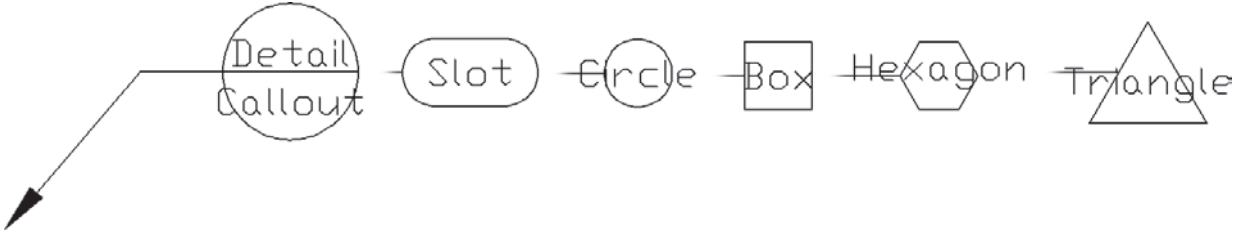


Notes AutoCAD calls numbers inside the bubbles “tags,” but they actually are attributes. To edit them, double-click the bubble (which is a block). In the Edit Attributes dialog box, you can change the tag number to any other number or text.

You can change the bubble type through the **PROPERTIES** command. In the Block section, click the **Source block** droplist, and then choose another style of bubble, as illustrated below.



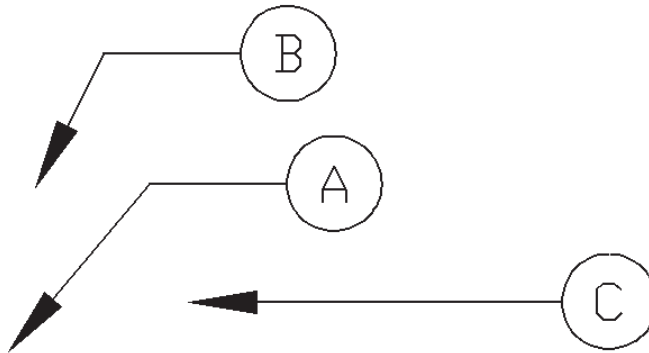
The predefined blocks (a.k.a bubbles) are illustrated below.



TUTORIAL: ALIGNING MLEADERS (MLEADERALIGN)

When numerous mleaders are in the same area of the drawing, they can look messy to some eyes. The **MLEADERALIGN** command lines them up nicely.

1. Draw two or more bubble mleaders following the steps described earlier.



2. To align multiple multiline leaders, start the **MLEADERALIGN** command:
 - From the Multileader toolbar, choose the **Align Multileaders** button.
 - In the ribbon's Annotate tab, choose the **Align** button in the Leaders panel.
 - At the 'Command:' prompt, enter the **MLEADERALIGN** command:
Command: **mleaderalign** (Press ENTER.)
 - Alternatively, enter the **mlla** alias at the 'Command:' prompt.
3. In all cases, AutoCAD prompts you to select two or more mleaders:

Select multileaders: **all**

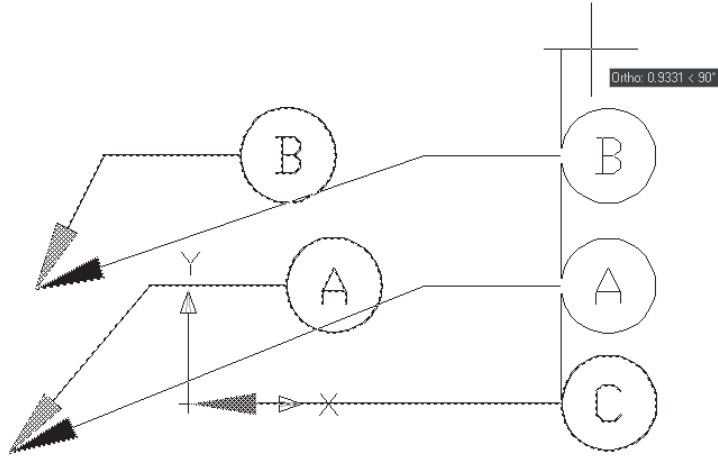
Select multileaders: (Press ENTER to exit object selection.)
4. AutoCAD asks you to pick one mleader; the others will align with it.

Current mode: Use current spacing

Select multileader to align to or [Options]: (Pick one multiline leader. In this tutorial, I've picked C.)

5. Pick a point to define the alignment. This is like drawing an xline:
Specify direction: *(Pick a point.)*

The screen gets messy as AutoCAD shows the mleaders in their original position and their current position. Notice that as you move the cursor, the bubbles of the other mleaders align themselves between the cursor and the base mleader.



Turn on ortho mode to align mleaders vertically or horizontally.

MLeaderAlign Options

This command has an **Options** option:

Enter an option [Distribute/make leader segments Parallel/specify Spacing/Use current spacing] <Use current spacing>:

Distribute — distributes the mleader content (bubbles or mtext) between two points. The distance and angle between the two points determine the spacing and angle of the content.

Specify first point or [Options]: *(Pick a point.)*

Specify second point: *(Pick another point.)*

make leader segments Parallel — makes the last leader segment of each mleader parallel. AutoCAD prompts you to select the one mleader to which the other leader lines are made parallel.

specify Spacing — allows you to specify the spacing between each leader.

Specify spacing <0.000000>: *(Enter a number, such as 0.5.)*

Select multileader to align to or [Options]: *(Pick one mleader.)*

Specify direction: *(Pick a point to indicate the angle.)*

Use current spacing — keeps the spacing of the leaders (default).

TEXTEDIT

The **TEXTEDIT** command edits text. (The name was changed in AutoCAD 2010 from **DDEDIT**, short for “dynamic dialog editor,” *dynamic dialog* being an old reference to dialog boxes.)

This command edits all types of AutoCAD text, whether single-line text, **mtext**, dimensions, fields, or attributes. For each type of text, however, it displays a different user interface.

TUTORIAL: EDITING TEXT

1. To edit text in drawings, start the **TEXTEDIT** command:
 - Double-click the text. (This action by-passes the ‘Select annotation to edit’ prompt.)
 - At the ‘Command:’ prompt, enter the **textedit** command:
Command: **textedit** (Press ENTER.)
 - Alternatively, enter the **ed** alias at the ‘Command:’ prompt.
2. In all cases, AutoCAD prompts you to select the text:
Select an annotation object or [Undo]: *(Select another line of text, or press ESC to exit the command.)*

AutoCAD provides different text editors, depending on the text you selected:

- The Inplace editor for single-line text placed with the **TEXT** command.
- The Text Formatting toolbar or ribbon for multi-line text placed with the **MTEXT** command, as well as for text in fields, tables, leaders, and dimensions.
- The Edit Attribute Definition dialog box for attribute definitions placed with the **ATTDEF** command, as well as for text in mleader bubbles.



Notes As an alternative to entering the **TEXTEDIT** command, you can double-click single-line and multiline text, and attribute definitions. AutoCAD automatically displays the appropriate editor.

Double-clicking leaders and dimension text displays the Properties window, which can also be used to edit text.

Attributes placed with the **INSERT** command are edited with a separate command, **ATTEDIT**.

Single-line Text Editor

When you select single-line text placed by **TEXT**, AutoCAD displays the inplace editor. Each line of text is edited independently of the others. See the **TEXT** command at the beginning of this chapter.

You can edit a group of text blocks by holding down the **ALT** key, and then selecting each text block.

When done editing, press **ENTER**. If you want to discard the editing changes, press **ESC**.

Multiline and Leader Text Editor

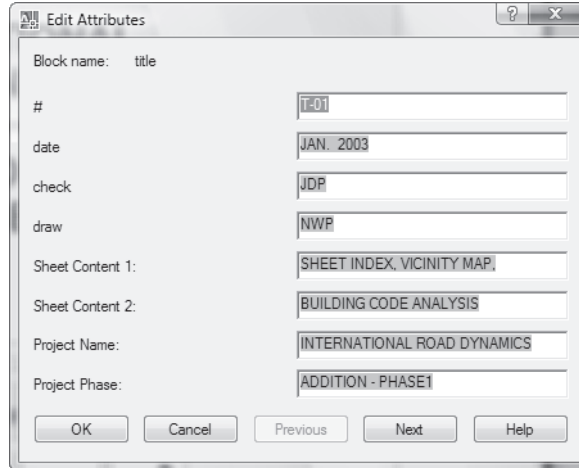
When you select leaders, dimensions, or text created by the **MTEXT** command, AutoCAD displays the Multiline Text tab (or Text Formatting toolbar) — identical to the one displayed by **MTEXT**.

Make your changes, and choose the **Close Text Editor** button. AutoCAD returns to the prompt:

Select an annotation object or [Undo]: *(Select another paragraph of text, or press ESC to exit the command.)*

Attribute Text Editor

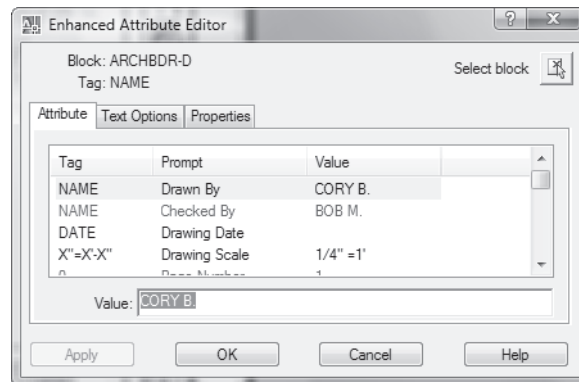
When you select attribute definitions created by the `ATTDEF` command, AutoCAD displays the Edit Attribute Definition dialog box. (Attributes are not discussed in this book.)



Make your changes to the tag, prompt, and default. Choose the **OK** button.

Enhanced Attribute Editor

When you select a block containing attributes, AutoCAD displays the Enhanced Attribute Editor dialog box. This is the same dialog box displayed by the `EATTEDIT` command.



Make your changes to the attributes, and then choose the **OK** button.

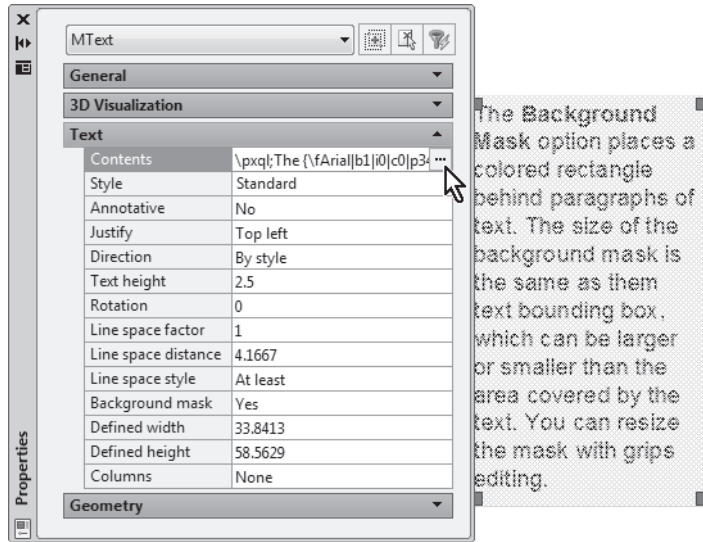
EDITING TEXT: ADDITIONAL METHODS

An alternative to the `TEXTEDIT` command is the `PROPERTIES` command. It displays a palette that allows you to edit text as well as to change many text properties — color, layer, insertion point, the background mask, and so on. The only property that the Properties palette cannot change is the font, which must be changed with the `STYLE` command.

To access this palette, select the text, and then right-click. From the shortcut menu, select **Properties**. AutoCAD displays the Properties palette.

The **Contents** area displays the text of the text object. When you see strange characters — such as `\L` and `\P` — these are mtext formatting codes.

If the text is paragraph text, choose the **[...]** button, which causes AutoCAD to open the familiar mtext editor.



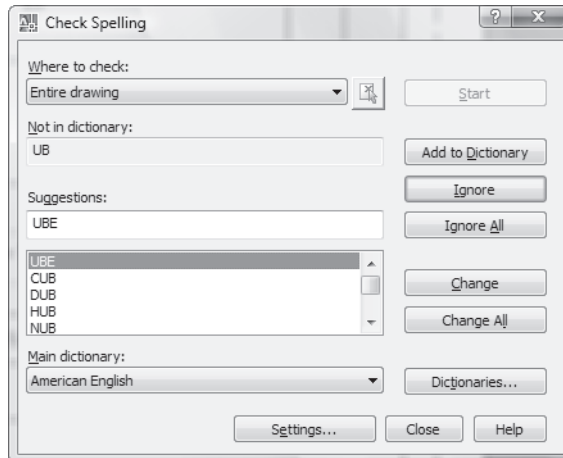
ABC SPELL

The **SPELL** command checks the drawing for words unfamiliar to AutoCAD.

TUTORIAL: CHECKING SPELLING

- To check the spelling of text, start the **SPELL** command:
 - In the ribbon's Annotate tab, click the **Check Spelling** button in the Text panel.
 - At the 'Command:' prompt, enter the **SPELL** command:
Command: **spell** (Press ENTER.)
 - Alternatively, enter the **sp** alias at the 'Command:' prompt.

In all cases, AutoCAD displays the Check Spelling dialog box.



- Click **Start** to begin checking the spelling of words.
- Confirm the spelling of words AutoCAD does not recognize.
When done, AutoCAD reports: "Spelling check complete."

This command works with words placed or imported by the **TEXT**, **-TEXT**, **MTEXT**, **-MTEXT**, **LEADER**, **QLEADER**, **MLEADER**, and **ATTDEF** commands. When it comes to attributes, values are spell checked, but not tags. Because **'SPELL'** can be run transparently, you can use it while other commands are active.



FIND

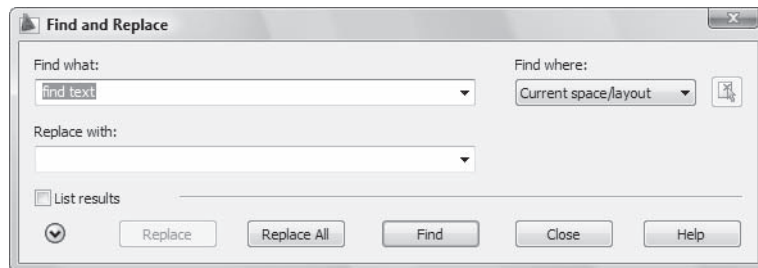
The **FIND** command searches for, and optionally, replaces text.

TUTORIAL: FINDING TEXT

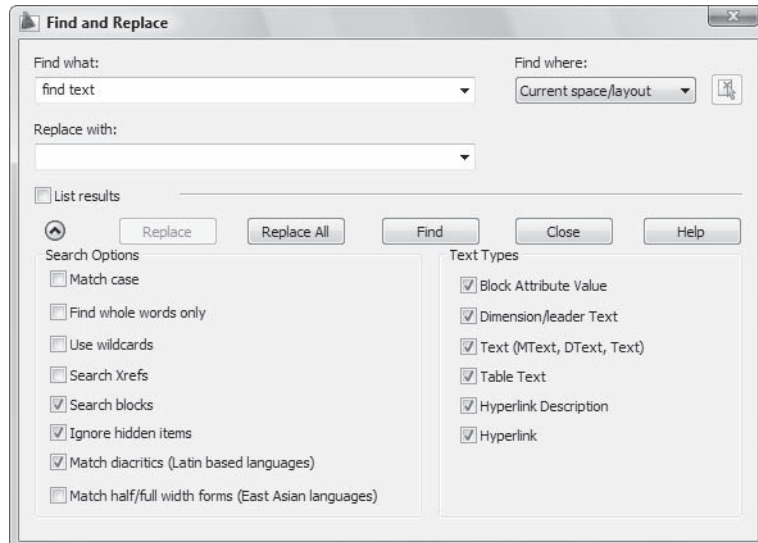
- To find and replace text, start the **FIND** command:
 - In the ribbon's **Annotate** tab, enter text in the **Find Text** field of the Text panel.
 - At the 'Command:' prompt, enter the **FIND** command.

Command: **find** (Press **ENTER**.)

- In all cases, AutoCAD displays the Find and Replace dialog box.



- Enter a word or phrase to search for in the drawing.
- To narrow the search, click **More**.



The expanded dialog box lets you narrow the search to specific kinds of words, such as those found only in dimensions or in table text. Field text is considered a form of mtext.

- Optionally, enter a replacement word or string.
- Click **Find Next** to find the first instance of the phrase.
If necessary, click **Replace** or **Replace All**.

7. When done, click **Close**.

The **U** command reverses changes made by this command.

SCALETEXT AND **JUSTIFYTEXT**

The **SCALETEXT** and **JUSTIFYTEXT** commands change the size and justification of text.

After placing text in drawings, you may find it necessary to change the height of the text, or its justification.

AutoCAD provides the **SCALETEXT** command to change the height (scale) of text. It makes selected text larger or smaller. The advantage of this command over **PROPERTIES** or **SCALE** is that it resizes multiple text objects without changing their location; the **PROPERTIES** command may not do it accurately, while the **SCALE** command scales everything, including location, relative to the base point.

The **JUSTIFYTEXT** command changes the justification of text. It can, for example, change left-justified text to right-justified. The style of justification of mtext can be changed with the mtext editor through the **MTEDIT** command.

TUTORIAL: RESIZING TEXT

1. To resize text, start the **SCALETEXT** command:
 - At the 'Command:' prompt, enter the **SCALETEXT** command.

Command: **scaletext** (Press **ENTER**.)

2. In all cases, AutoCAD prompts you to select text.

Select objects: (Pick one or more lines of text.)

Select objects (Press **ENTER** to end object selection.)

3. Pick a base point, or select a justification point:

Enter a base point option for scaling [Existing/Align/Fit/Center/Middle/Right/TL/TC/TR/ML/MC/MR/BL/BC/BR] <Existing>: (Enter an option, or press **ENTER**.)

4. Specify the new height:

Specify new height or [Match object/Scale factor] <3/16">: (Enter a height, or an option.)

When AutoCAD prompts you to select objects, you can choose a mix of text and non-text objects; AutoCAD filters out the non-text objects automatically.

The **SCALETEXT** command changes the height of selected text relative to a *base point*. The base point can be the existing insertion point or one of the many text justification points. By default, the base point is the existing insertion point.

The **Match Object** option matches the height to that of another text object:

Select a text object with the desired height: (Pick another text object.)

Height=3/16"

The **Scale Factor** option scales the text by a factor. A factor larger than 1 enlarges the text, while a value under 1 reduces it.

Specify scale factor or [Reference]: (Enter a factor, such as **2**.)

The **Reference** option is identical to that of the **SCALE** command: the text is scaled relative to another size.

TUTORIAL: REJUSTIFYING TEXT

1. To change the justification of text, start the **JUSTIFYTEXT** command:
 - At the 'Command:' prompt, enter the **JUSTIFYTEXT** command.
 Command: **justifytext** (Press ENTER.)
2. In all cases, AutoCAD prompts you to select text.
 Select objects: (Pick one or more lines of text.)
 Select objects (Press ENTER to end object selection.)
3. AutoCAD prompts you to select a justification option; the option you select will override the justification of the existing text.
 Enter a justification option [Existing/Align/Fit/Center/Middle/Right/ TL/TC/TR/ML/MC/MR/ BL/BC/BR] <Existing>: (Press ENTER.)

The same rules apply to the **JUSTIFYTEXT** and **SCALETEXT** commands regarding object selection. The **U** command can undo any damage.



ANNOTATION SCALING

Annotation scaling ensures only correctly-scaled objects appear in Model tab and model space viewports.

"Annotation" refers to objects that annotate drawings. (*Annotation* means "explanatory text.") In AutoCAD, this includes the following objects:

Single-line text	Mtext	Field text
Attribute text	Dimensions	Tolerances
Multiline leaders	Hatch patterns	Linetypes

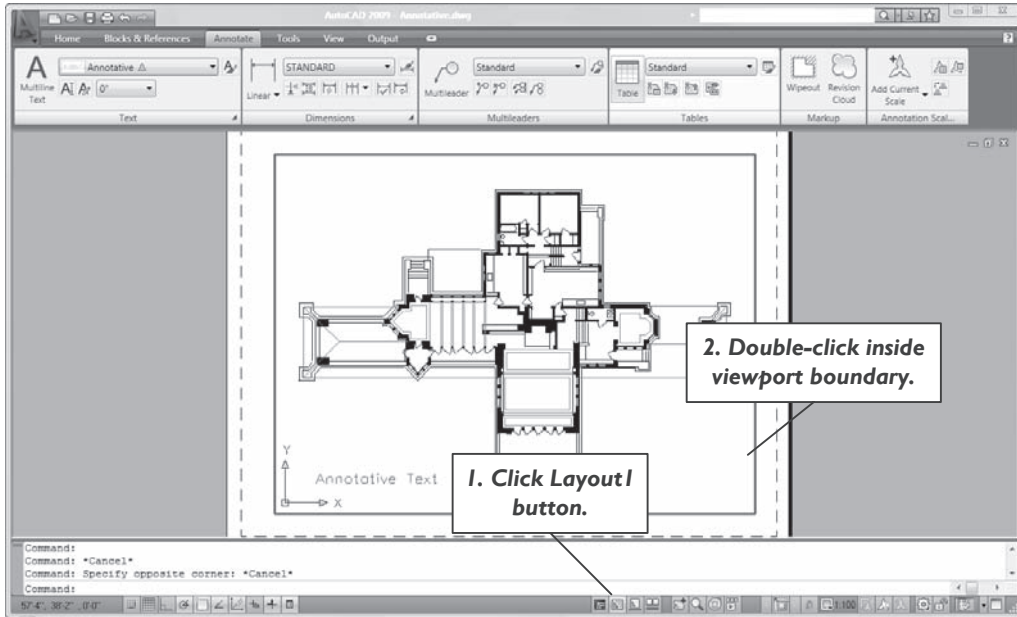
As described earlier in this chapter, text needs to be scaled larger to appear at the correct size when the drawing is plotted smaller to fit the paper or vice-versa. Annotation scaling is a system devised by Autodesk to solve the problem semi-automatically. It works like this:

A layout viewport has its scale factor set to **1:100**. Draw text with an annotative style.
 AutoCAD automatically assigns the text the inverse scale factor of **100:1**.

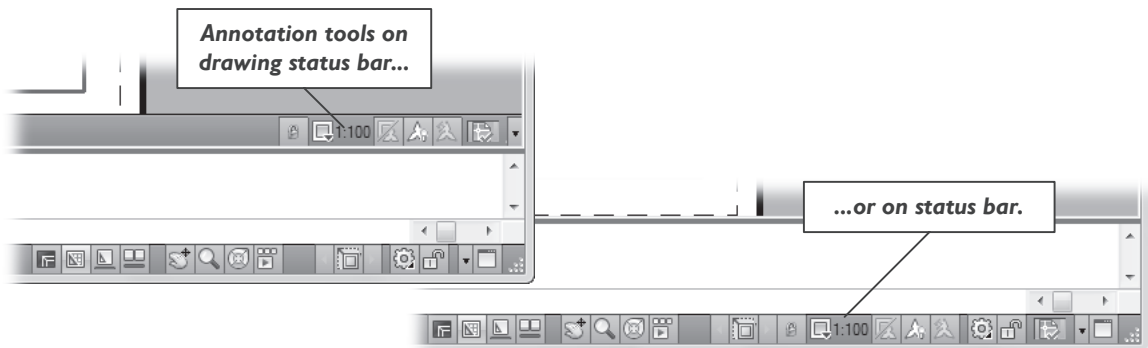
There is no "Annotation" command; instead, annotation is a property integrated into some commands, styles, and user interface elements. Let's work through a tutorial, and then examine the additional options.

TUTORIAL: AUTOMATICALLY SCALING TEXT

1. In AutoCAD, open the *Annotative.dwg* drawing file from the Companion CD.
2. Switch to layout mode by clicking the **Layout1** tab or button.
3. Enter model space by double-clicking inside the viewport border.



Notice the tools for working with annotative scaling. These can appear in two places, depending on how AutoCAD is set up: one possible location is on the drawing bar (above the command bar); the other is on the status bar (below the command bar).



(The location is switched through the **OPTION** command's **Display Drawing Status Bar** option in the Display tab.)

4. The drawing is not correctly scaled in the viewport; it should be scaled to meet two criteria: (1) the entire drawing must fit the viewport, and (2) the drawing must be at a standard scale factor. To select an appropriate scale factor, follow these steps:
 - a. Use the **zoom Extents** command to fit the drawing to the viewport.
 - b. Notice the scale factor next to **VP Scale**: it reads 0.010998 or something similar.
 - c. Click **VP Scale**, and then select the nearest standard scale factor of **1:100**. Notice that the drawing reacts by zooming slightly smaller, and that the Annotation Scale factor matches.



Notes The **VP Scale** button displays a long list of scale factors. You can edit this list with the **SCALELISTEDIT** command, removing and adding scale factors. This single command affects the scale factors available to viewport scaling, plot scaling, and annotation scaling.

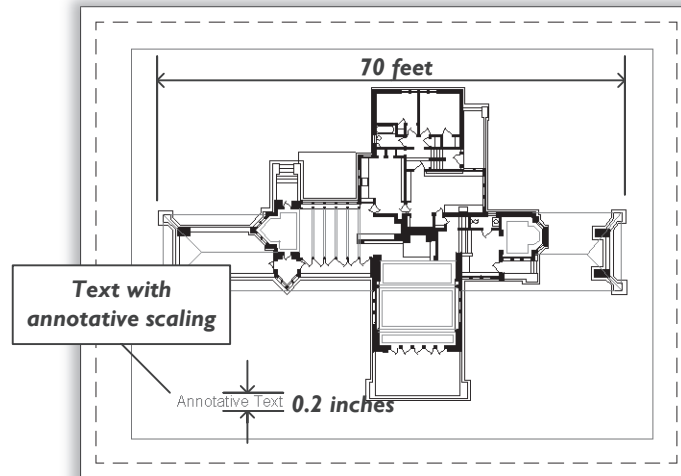
To reduce the list further, set the **HIDEXREFSCALES** system variable to 1, which hides scale factors imported from attached xref files.

5. To test the annotation feature, place some text with the **TEXT** command. Follow these steps:
 - a. Switch to Model tab.

The technical editor explains why: “If you zoom and pan while entering text in a layout tab, you mess up the viewport’s scale. When you re-enter model space, the annotations are ‘missing,’ because their scale no longer matches the viewport scale. For this reason, it is best to enter annotative text in Model tab.”

- b. From the Dashboard’s Text panel or the Styles toolbar, select the Annotative text style.
- c. Start the **TEXT** command.
Command: **text**
- d. Notice that the prompt reports that the annotative property is turned on:
Current text style: "Annotative" Text height: 0'-0" Annotative: Yes
Specify start point of text or [Justify/Style]: *(Pick a point in the viewport.)*
- e. Notice that the prompt asks for “paper height”: this is the height of the text as you want it to appear when plotted. In this case, enter a height of 0.2", which is close to the ideal of 3/16":
Specify paper height <0'-0">: **0.2"**
Specify rotation angle of text <0>: *(Press ENTER.)*
Annotative Text *(Press ENTER twice.)*

Notice that the text appears at the correct size in the viewport. As illustrated below, the floorplan is 70 feet wide, yet the text is legible despite it being just 0.0017 feet tall (0.2").



6. To see why this is possible, use the **LIST** command, and then select the text. I’ve highlighted the important parts below:

TEXT	Layer: "0"
Space:	Model space
Handle =	5d5
Style =	"Annotative"
Annotative:	Yes
Annotative scale:	1:100
Typeface =	Arial
start point,	X= 9'-0" Y= -14'-6" Z= 0'-0"
paper text height	0'-0"
model text height	1'-8"
text	Annotative Text

The text contains the annotative scale property of 1:100, and then applies it to text in model space (“model text height”). We entered a height of 0.2". Behind the scenes, AutoCAD automatically inverted the scale to draw the text at $0.2" \times 100:1 = 20"$ (same as 1'-8" in feet-inches).

The **paper text height** is set to 0'0", even though you entered 0.2". That's because AutoCAD reverse-calculates the height from the annotative scale factor multiplied by the model text height. The paper text height changes automatically when the associated annotative scale changes.

7. To see the effect of annotative scaling, change the scale of the viewport. From the status bar, click **VP Scale** and select **1:50**. Notice that the text disappears. (If necessary, pan the drawing to where the text is/was located.)
8. One of the status bar buttons lets you see annotative-scaled objects, no matter what the viewport scaling is. Click the  **Annotation Visibility** button. (This button toggles the **ANNOALLVISIBLE** system variable.) Notice that the text reappears and is twice as large.
9. When you work with several viewports scales, it can be a pain to assign annotative scales to a large numbers of objects. To have AutoCAD automatically assign the scale factors, follow these steps:
 - a. Click the  **Automatically Add Scales** button. (This button toggles the **ANNOAUTOSCALE** system variable.)
 - b. Change **VP Scale** to something else, such as **1:40**. Notice that the text does not become larger (as you might expect from step 7), but returns to its former size.
 - c. Change **VP Scale** to **1:50**. Again, the text is the “correct” size. No matter which viewport scale you now select, the annotatively-scaled text appears at constant size.



Notes When objects have more than one annotative scale, the selection cursor shows a double triangle icon, as illustrated below



Autodesk warns against objects having too many annotative scale factors, saying that this could slow down your computer. Curiously, the **LIST** and **PROPERTY** commands do not list multiple scale factors; instead, you need to use the **OBJECTSCALE** command to view, add, and remove scale factors from objects.

ANNOTATION SCALING: ADDITIONAL METHODS

While no single command controls annotative scales, these system variables and commands fine tune the system:

- **OBJECTSCALE** command edits annotative scales associated with objects.
- **SELECTIONANNODISPLAY** and **XFADECTL** system variables dim the display of annotative objects whose scale differs from the viewport scale.
- **ANNOUNUPDATE** command updates objects to match a recently-changed style.
- **ANNORESET** command forces the scale and visibility to match the current settings.
- **ANNOTATIVEDWG**, **DIMANNO**, and **MSLTSCALE** system variables control the annotative properties of drawings inserted as blocks, dimensions, and linetypes, respectively.
- **SAVEFIDELITY** system variable determines whether drawings are saved with visual fidelity to earlier releases of AutoCAD.

Let's look at each.

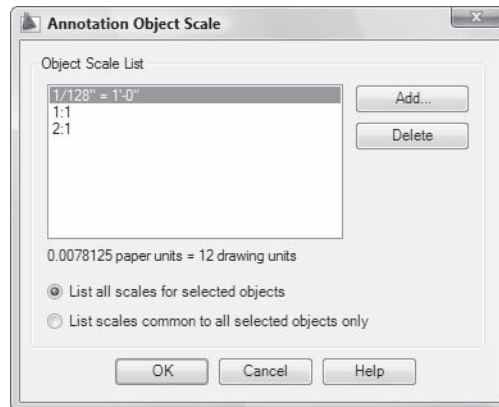


The OBJECTSCALE command edits annotative scales associated with objects through a dialog box.

Command: **objectscale**

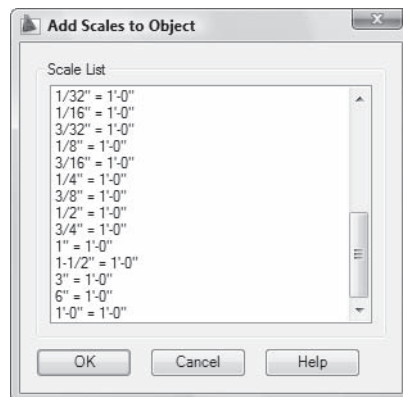
Select annotative objects: *(Pick one or more objects.)*

Select annotative objects: *(Press ENTER to end object selection.)*



The dialog box lists the scale factor(s) assigned to the selected object(s).

Add adds scale factors to the objects, in a roundabout manner; displays the following dialog box:



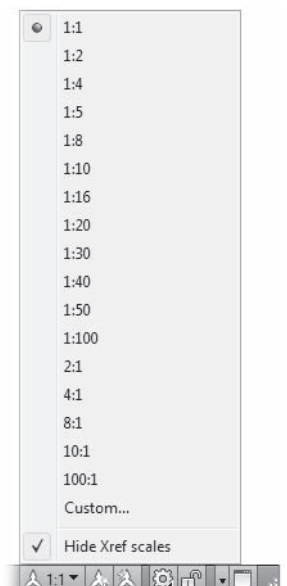
This dialog box copies the list maintained by the SCALELISTEDIT command, which also handles scale factors for viewport and plot scaling; if you don't see the scale factor you need, then you need to use the SCALELISTEDIT command to add it, as detailed in Chapter 20 on plotting.

Delete deletes scale factors. You can select one or more for deletion, but you cannot delete all of them, because at least one annotative scale must remain. (To remove the last remaining annotative scale, use the PROPERTIES command to turn off the object's Annotative property.)

The following options determine which scale factors are listed by the Annotation Scale Factor dialog box:

☒ **List All Scales For Selected Objects** — displays all scales factors from all the objects you selected.

☐ **List Scales Common to All Selected Objects Only** — displays only the scale factors that the selected objects share.



SelectionAnnoDisplay and XFadeCtl

The **SELECTIONANNODISPLAY** system variable toggles the display of annotative objects whose scale differs from the viewport scale.

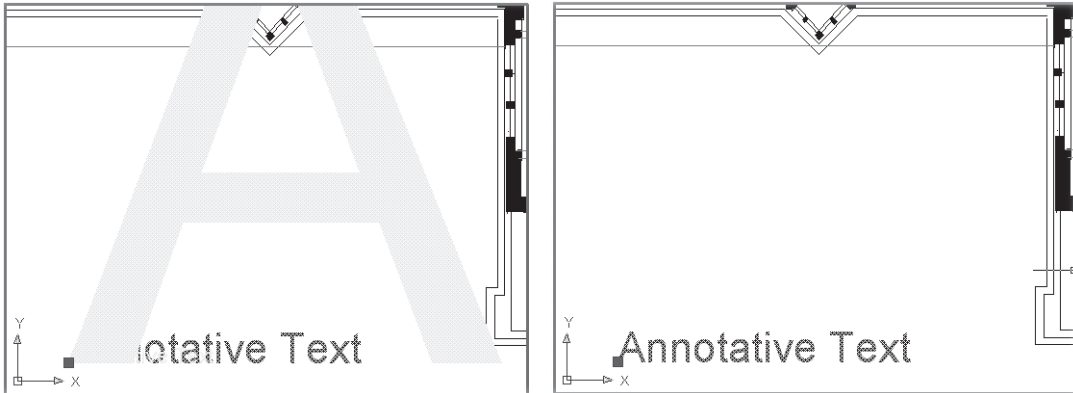
Command: **selectionannodisplay**

Enter new value for SELECTIONANNODISPLAY <1>: (Type 1 or 0.)

0 — Only the annotative objects that match the viewport scale are displayed.

1 — All annotative objects are displayed, with non-current ones faded (default).

At left in the figure below, **SELECTIONANNODISPLAY** is on. The annotative text is selected and displays its multiple-scale representations; those not at the current viewport scale are faded (light gray).



Left: **SELECTIONANNODISPLAY** = 1, **XFADCTL** = 90.

Right: **SELECTIONANNODISPLAY** = 0. (Value of **XFADCTL** doesn't matter.)

At right, **SELECTIONANNODISPLAY** is off, and so only the representation that matches the viewport scale is displayed.

The level of dimming is controlled by a second system variable, **XFADCTL** (short for “xref fade control”). The level is a percentage between 0 and 90:

Command: **xfadctl**

Enter new value for XFADCTL <50>: (Enter a value between 0 and 90.)

0 — Off, no dimming.

90 — Maximum dimming.



Notes When annotative text has multiple representations, you can move each independently of the others, as illustrated below. To do this, set a viewport scale that matches one representation, and then move it; select the next viewport scale, and so on.



While you can move multiple representations independently of each other, you cannot change properties, such as color, independently.

The **DRAWORDER** command has no effect on changing the display order when multiple representations overlap, because they all represent the same text, albeit at different sizes.

AnnoUpdate

The **ANNOUPDATE** command updates objects to match a recently-changed style.

Command: **annoupdate**

n found *n* were updated

You use this command to update annotative objects when the related styles are changed, such as text and dimension styles.

AnnoReset

The **ANNORESET** command forces the location of scale representations to match the current settings.

Command: **annoreset**

Reset alternate scale representations to current position

Select objects: *(Pick one or more objects.)*

Select objects: *(Press ENTER to end object selection.)*

WHERE ANNOTATIVE SCALING HAPPENS

Annotative scaling is scattered about AutoCAD in many nooks and crannies. It is a property that can be set in styles and commands, and changed after the fact with these commands and system variables. Annotative styles are indicated in lists by the triangle icon:  Annotative

Style Commands

DIMSTYLE command — affects dimensions and tolerances.

MSTYLELEADER command — affects multiline leaders.

STYLE command — affects single-line text, mtext, and fields.

Drawing Commands

ATTDEF and **BATTMAN** commands — create and edit attributes.

INSERT command — reports annotatively-scaled blocks and dynamic blocks.

HATCH and **HATCHEDIT** commands — draw and edit hatches.

TEXT, **MTEXT**, and **TEXTEDIT** commands — draw and edit text, mtext, and fields.

MSLTSCALE system variable — scales all linetypes annotatively.

Properties Command

The **PROPERTIES** command changes the annotative property for single-line text, mtext, fields, dimensions, hatches, attributes, multileaders, leaders, qleaders, and tolerances. Typical annotation-related properties are shown below for text.

Text	
Contents	Annotative Text
Style	 Annotative
Annotative	Yes
Annotative scale	1:50
Justify	Left
Paper text height	0"
Model text height	10"
Match orientation to layout	No

ANNOTATION BAR

There is no command for controlling annotation scaling, because it is a property. You control it through the “annotation bar” (illustrated below), or through system variables, properties, some editing and style commands, and the Properties palette.

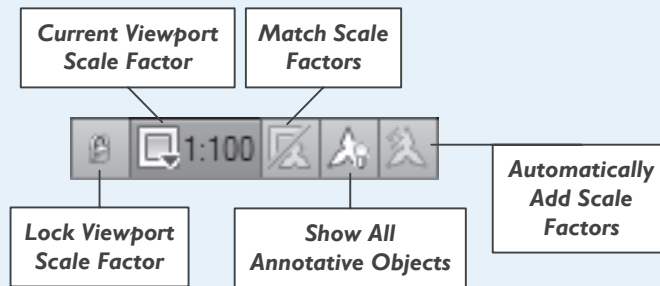
In Model tab, these annotation controls are displayed on the status bar:



In Layout tab, only two controls are displayed when in paper space:



When in a layout’s model space, the following controls are shown:



Lock Viewport Scale Factor — toggles locking of the viewport scale; also locks the annotation scale. This option is also available through the viewport’s properties.

Current Viewport Scale Factor — reports the scale of the current viewport; click the arrow to select another scale factor.

Match Scale Factors — makes the viewport and annotative scale match.

Show All Annotative Objects — toggles the display of annotative objects:

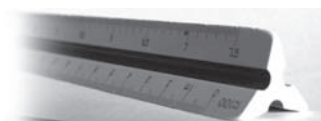
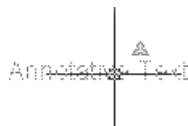
- On** — all objects are displayed, regardless of their annotative scale.
- Off** — only objects whose annotative scale matches the viewport scale are displayed.

Automatically Add Scale Factors — toggles addition of annotative scales to objects:

- On** — when the viewport scale changes, the new scale factor is added to annotative objects automatically.
- Off** — objects keep their annotative scale factors fixed.



Note AutoCAD identifies objects with annotative scaling through the triangular icon . As you pass the cursor over objects, they highlight, and those with annotative scaling display the icon near the cursor.



(History: The icon represents the end view of a triangular scale ruler used by architects and engineers. Being triangular in cross-section, it provided 12 scale factors, two along each of the six sides. The technical editor notes that these rulers are excellent for firing elastic bands at coworkers.)

The technical editor also notes that this command does not reset annotatively-scaled objects to their original locations, but instead moves them to match the current location of the one used to trigger the change.

AnnotativeDwg

The ANNOTATIVEDWG system variable controls the annotative property of drawings inserted as blocks into other drawings. It is read-only when the drawing contains at least one annotative object; otherwise, you can change the value between 0 and 1.

Command: **annotativedwg**

ANNOTATIVEDWG = 0 (read only)

0 — Behaves nonannotatively.

1 — Behaves annotatively.

DimAnno

The DIMANNO system variable reports whether the current dimension style has the annotative property.

Command: **dimanno**

DIMANNO = 1 (read only)

0 — dimension style is not annotative.

1 — dimension style is annotative; use the DIMSTYLE command to change the annotative property, which affects the size of dimension text and arrows. See Chapter 15, “Editing Dimensions.”

MsLtScale

The MSLTSCALE system variable toggles the annotative property of linetypes in model space (short for “model space linetype scale”).

Command: **msltscale**

Enter new value for MSLTSCALE <0>: (Type 1 or 0.)

0 — linetypes are not scaled by the annotation scale; the default for drawings imported from AutoCAD 2007 and earlier.

1 — linetypes are scaled by the annotation scale; the default for drawings created in AutoCAD 2008 and later.

This system variable does not come into effect until after the next drawing regeneration. (Use the REGENALL command, if necessary.) To use this system variable effectively, all other linetype scale-related system variables should also be set to 1:

LtScale = 1 or a scale correct for plotting.

CeLtScale = 1

PsLtScale = 1

SaveFidelity

The SAVEFIDELITY system variable determines whether drawings with multiple annotatively-scaled objects are saved with *visual fidelity*. Earlier releases of AutoCAD cannot handle these objects, and so visual fidelity ensures that they are saved to separate layers

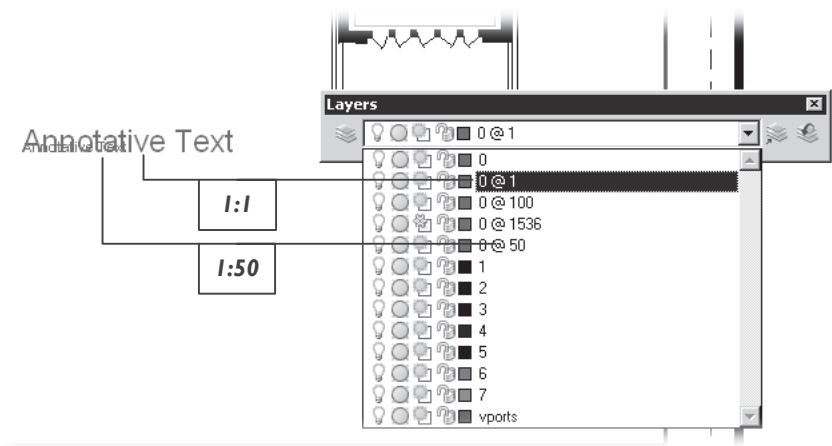
Command: **savefidelity**

Enter new value for SAVEFIDELITY <1>: Type 1 or 0.)

0 — Drawings save only annotative objects at the current scale, and discard the others.

1 — Drawings save each scaled representation of annotative objects on separate layers (default).

The problem with multiple representations of annotatively-scaled text is that they display correctly only in AutoCAD 2008 and later. This system variable determines what to do when drawings are opened in earlier releases of AutoCAD.



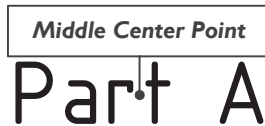
When set to 1 (the default), drawings contain extra layers whose names are marked with the scale factor, such as “0 @ 50,” as illustrated above. This layer contains all annotative objects found on layer 0 bearing the scale factor of 1:50. The layer is frozen when first opened in AutoCAD 2007 (or earlier), and can be thawed to view the text. Everything returns to normal when re-opened in 2008 or later.

EXERCISES

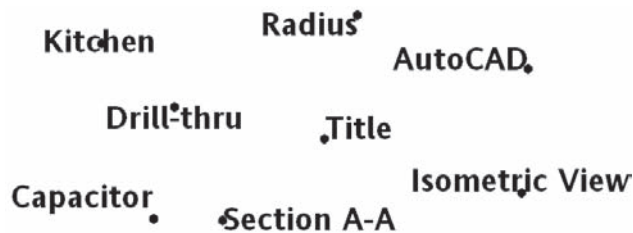
1. In this exercise, you practice placing text.
Start AutoCAD with a new drawing. With the **TEXT** command, place following lines of text:

THIS IS THE FIRST LINE OF TEXT
THIS IS THE SECOND & CENTERED LINE OF TEXT

2. In this tutorial, you place text centered on a point.
The text “Part A” is to be centered both vertically and horizontally on a selected point. Use the **TEXT** command to place the text. Which justification mode did you use?



3. In this exercise, you place text with a variety of alignments.
The figure illustrates several text strings. The placement point is marked with a solid dot. Use the **TEXT** command to place the text as shown.



4. In this exercise, you place the following line of text at a variety of angles:
This text is rotated
Using the **TEXT** command, place the following line of text at the following angles:
 - a. 0 degrees.
 - b. 45 degrees.
 - c. 90 degrees.
 - d. 135 degrees.
 - e. 180 degrees.
 Use the same base point for each line.
5. In this exercise, use control codes to construct the following text string in the drawing.
The story entitled MY LIFE is ±50% true.
6. In this exercise, you practice using mtext. Start a new drawing, and use the **MTEXT** command to place several lines of text in the drawing, such as your name and address.
Your First and Last Name
1234 First Avenue
Anytown, BC
V8C 1T2 Canada

Click the **OK** button to exit the mtext editor.

7. Continuing from the previous exercise, double-click the mtext. Does the mtext editor reappear?
 Make the following changes to parts of the text:
 Font: **Times New Roman**
 Color: **Red**
 Click the **OK** button to exit the mtext editor. Did the changes come into effect?
8. Continuing from the previous exercise, turn on the **QTEXT** command.
 Did the text change its look?
 If no, which command did you forget to use?
9. In this exercise, you import text from another file.
 If necessary, create a brief text file in Notepad, the text editor included with every version of Windows. Save the file in .txt format.
 Import the text file into your drawing with the **MTEXT** command.
10. Create five different styles that represent very different text appearances, such as wide text, slanted, and so on.
 Which command creates styles?
 Place examples of each style in the drawing.
11. Using only the **MTEXT** command, design your own business card with at least two fonts, three font variations, and two colors.
 Do not design a logo — just position the text. Include at least your name, address, and Internet information.
 Standard business cards are rectangles that measure 3.5" wide by 2" tall.
 Which command is useful for drawing rectangles?
 How many business cards can you fit onto a vertical A-size sheet?

 (The technical editor quotes Lynn Allen: you know you've been working with AutoCAD too long when you use it to design wedding invitations.)
12. Use a combination of text and drawing commands to design a logo for an engineering office. The logos often include variations on the owners' names.
13. In this and the next exercise, you practice drawing leaders.
 Draw a rectangle.
 Attach a leader to each of the four corners.
 Use the following text for each leader:
 North East Corner
 North West Corner
 South East Corner
 South West Corner
14. Continuing from the previous exercise, place a splined leader pointing to the center of the rectangle. Change the arrowhead to a dot.
 Use the following annotation:
 Parcel of Land.

15. Continuing from the previous exercise, double-click the annotation of the splined leader, and change the text to:

Disputed Parcel of Land.

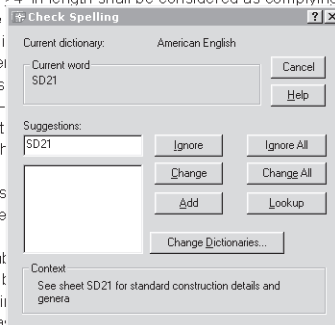
Not to be Subdivided.

16. In this and the following exercise, you correct and change text in drawings.

From the Companion CD, open the *spelling.dwg* drawing file, a drawing that contains text with spelling errors.

SPELL CHECKING EXERCISE

1. See sheet SD21 for standard construction details and general structural notes.
2. Sheath all specified interior walls per plans and "Structural Wall Sheathing Schedule" on sheet SD1. All exterior wall panels >4' in length shall be considered as complying braced wall panels per section 2320.11.3.
3. Install sheets with face stagger sheets and nail
4. For truss shapes, dime
5. All trusses shall be des
6. Contractor shall submit Standard ANSI/TP' 1 1- licensed in the State wr placement plan. These truss documents reviewed by the Engine working drawings. Trusses shall not be fat
7. All roof openings shall t
8. Provide full bearing at ii Use hardwood shims a
9. Provide multiple stud and/or solid blocking under all multiple trusses.
10. Truss Designer shall design gable end trusses for wind loads perpendicular to face of truss.



Correct the spelling.

How many mistakes does AutoCAD's **SPELL** command find?

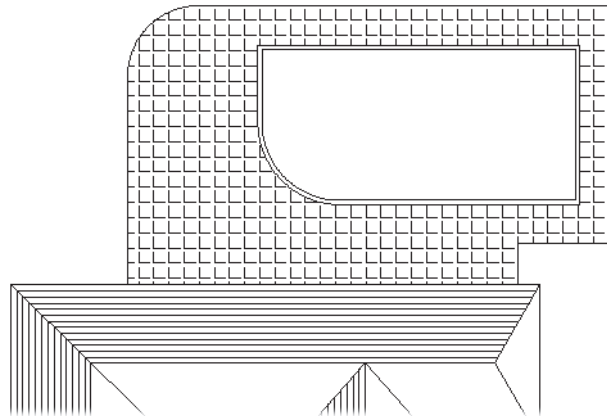
How many *real* spelling mistakes are there?

17. Continuing with the same drawing, find and replace the following phrases:

Find	Replace With
sheet SD1	sheet AA-01
Building Official	local Building Official

Which command did you use to find and replace words in drawings?

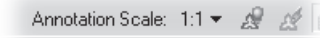
18. From the Companion CD, open the *insert.dwg* drawing file, a house plan.



Use the **MTEXT** command to place the following note over a white rectangle on the patio area at the rear of the house. Use the background mask feature.

Interlocking Belgium Bricks

19. Continuing with the same drawing, use the **SCALETEXT** command to reduce the size of the text you added in the previous exercise.
20. Continuing with the previous exercise, use the **JUSTIFYTEXT** command to change the justification to **Center**. How does the text change?
21. In this exercise, we see the effect of using the following buttons on annotative text:



From left to right, these are:

- Annotation Scale (**CANNOSCALE** system variable) — changes the annotation scale.
 - Annotation Visibility (**ANNOALLVISIBLE** system variable) — toggles visibility of text at all annotation scales.
 - Add Annotation Scales Automatically (**ANNOAUTOSCALE** system variable) — toggles addition of scales to objects as annotation scale changes.
- a. Start a new drawing, and then turn off all of the settings, as listed below. You can use the controls on the status bar, or change the values of the system variables.

Annotation Scale	1:1	(CANNOSCALE = 1:1).
Annotation Visibility	off	(ANNOALLVISIBLE = 0)
Add Annotation Scales Automatically	off	(ANNOAUTOSCALE = -4)

Creating an Annotative Text Style

- b. Create a new text style named "Anno" with the following properties:

Annotative	☒ (on)
Paper Text Height	0.25
Font Name	Arial

Set the style current, and then close the dialog box.
- c. With the **TEXT** command, place the following text in the drawing in model space:

Annotative Text = 0.25
- d. With the **RECTANGLE** command, draw a rectangle that encompasses the extent of the text, as illustrated below. This rectangle will help you see the size and location of the original text as the annotation scale changes.

Annotative Text = 0.25

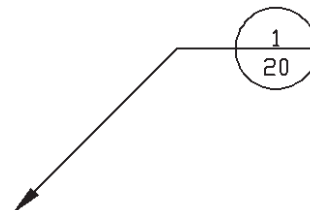
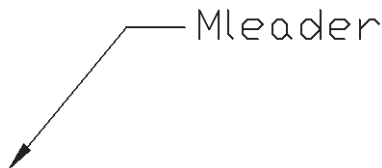
- e. Change **Annotation Scale** to **1:2**.
 What happens to the text?
 Why did the text change in that way?

Controlling Annotation Visibility

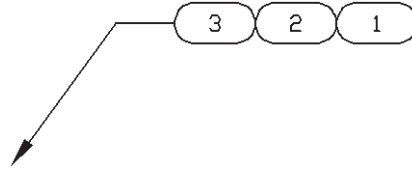
- f. Click the **Annotation Visibility**  button (or set **ANNOALLVISIBLE** = 1).
Why does the text appear?
- g. Change **Annotation Scale** to **1:4**,
Click the **Add Annotation Scales Automatically**  button (or set **ANNOAUTOSCALE** = 4).
Why does the text change its look?
- h. Change **Annotation Scale** back to **1:2**, and then select the text.
What do you see on the screen?

Resetting and Updating Annotative Text

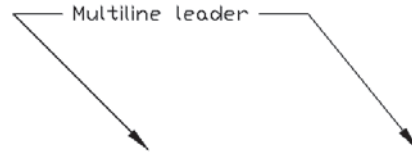
- i. With the text still selected, use its grip to move it a few inches away.
What happens to the other representations of the text?
 - j. With the text still selected, use the **ANNORESET** command.
To see the effect of the command, select the text.
What happens to the other representations of the text?
 - k. Return to the **STYLE** command, and change the **Paper text height** to **0.5**.
Click **Apply** and then **Close**.
What happens to the text?
 - l. Select the text, and then enter the **ANNOUPDATE** command.
What happens to the text?
 - m. Select the text.
What happens to the other representations?
(Keep this drawing open for the following exercise.)
22. In this exercise, you create *oriented* text. Continue with the drawing from the previous exercise.
- a. Start the **STYLE** command, and then turn on the **Match Text Orientation to Layout** option for the “Anno” text style.
Click **Apply** and then **Close**.
 - b. Use the **ANNOUPDATE** command to update the style of the text in the drawing.
 - c. Use the **ROTATE** command to rotate all objects in the drawing by 90 degrees.
What happens to the annotative text?
23. Use the **MLEADER** command to create the following leaders:
- a.
 - b.



- c. Draw the following multi-bubble mleader. Which command did you use to create it?



- d. Open the *mleader.dwg* file from the Companion CD. Add one more leader to make it look like the following figure. Which command did you use to add the leader?



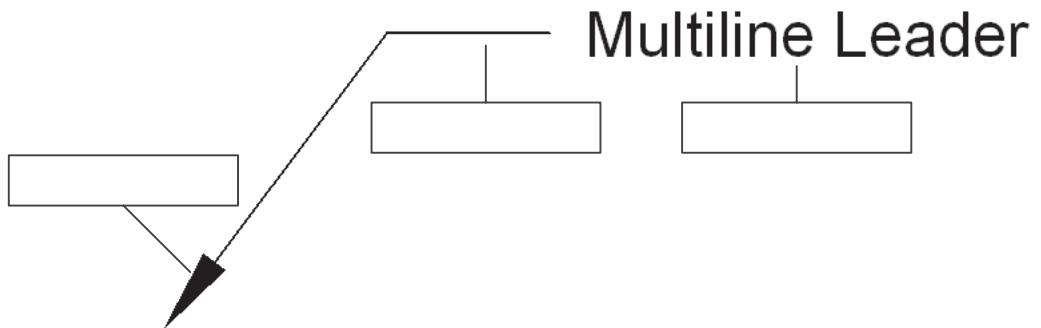
CHAPTER REVIEW

1. Name three ways you can see a list of the text styles stored in the drawing.
 - a.
 - b.
 - c.
2. What are *fonts*?
How do styles differ from fonts?
3. What four text properties are altered by the **STYLE** command?
 - a.
 - b.
 - c.
 - d.
4. There are several methods of placing text. List three of them, with a brief explanation of each:
 - a.
 - b.
 - c.
5. When an underscore or overscore is added to text, is the entire string altered, or can words be treated separately?
Explain.
6. When AutoCAD asks for a text height, is a numerical entry required?
Explain.
7. List five alignment modes available through the **TEXT** command.
 - a.
 - b.
 - c.
 - d.
 - e.
8. Can text be rotated at any angle?
How does the oblique angle differ from the text angle?
9. Does annotative scaling apply to hatch patterns and dimensions?
10. How can the text height be altered each time you enter text without redefining the style?
11. How can you compress and expand the text width?
12. Which formats of text can AutoCAD import into drawings?
13. What height should note text be in drawings?
14. Name four areas where text is used in drawings:
 - a.
 - b.
 - c.
 - d.
15. Can AutoCAD use *any* TrueType font found on your computer?
16. What happens when you open a drawing that contains a font not found on your computer?
17. Which command is better for placing lines of text at many locations over the drawing:
TEXT or **MTEXT**?

18. List the command associated with each alias:
 dt
 t
 ed
 st
19. Can you access the menu while entering text?
20. What do the three horizontal lines of the I-beam cursor represent?
 a
 b
 c
21. Which one of the following letters has a descender?
 a
 B
 d
 j
 L
22. Explain the meaning of the following justification codes:
 TL
 MC
 BR
 TR
23. Is BR justification the same as the **Right** justification? _x
 Explain.
24. Describe the one difference between **Align** and **Fit** justification modes.
25. What height should text be for a drawing scaled at:
 1:100
 1" = 50'
 1:1
 1" = 6"
 Show your work for each calculation.
26. Why is it wrong to use 1/8"-high text in an A-size drawing of a house?
27. When might you place rotated text in drawings?
28. Explain the meaning of the following control codes:
 %%u
 %%%
 %%d
 %%c
29. If you fail to turn off underlining, what happens on the next line of text?
30. What does *string* mean?
31. If the Euro symbol is not on your keyboard, how would you enter it?
32. Can you use more than one font with the **TEXT** command?
 With the **MTEXT** command?
33. Explain the meaning of the following mtext editor keyboard shortcuts:
 CTRL+B
 CTRL+I
 CTRL+A

34. How does AutoCAD stack text on either side of these characters?
/
^
#
35. Can different colors be applied to text placed with the **TEXT** command?
With the **MTEXT** command?
36. Explain the meaning of the following zeros:
Text Height = 0
Mtext Width = 0
37. Can only numbers be used for stacked text?
38. Is it possible to use a text editor or word processor other than AutoCAD's built-in mtext editor?
If so, how?
39. List the three primary parts of a leader:
a.
b.
c.
40. What is the quickest way to edit the text of a leader?
To edit the position of leader line?
41. Can leaders use only one kind of arrowhead?
42. Name three kinds of leader annotation:
a.
b.
c.
43. What does a splined leader line look like?
44. Can leaders be attached to objects?
45. List three ways to edit text:
a.
b.
c.
46. Is a different command needed to edit text created by the **TEXT** and **MTEXT** commands?
47. Does a different editor display for text created by the **TEXT** and **MTEXT** commands?
48. Can you have more than one text style in drawings?
49. What happens when you apply a different style to a line of text?
50. What happens to text when you make changes to a style?
51. Which font most closely approximates the Leroy lettering guides?
52. Can TrueType fonts have the vertical style?
53. Name one advantage to using styles in drawings.
54. What is the purpose of the **TEXTFILL** and **TEXTQLTY** system variables?
When do they come into effect?
55. Can the **SPELL** command be used for text placed by commands other than **TEXT** and **MTEXT**?
56. Explain a benefit of the **MTEXT** command's background mask.
57. What happens to orientated text when it is rotated?
58. Describe the purpose of the **SCALETEXT** command.
And the **JUSTIFYTEXT** command.

- 59. How would you change the size of text in a drawing being converted from imperial to metric units?
- 60. How is annotative scaling applied to text?
- 61. How does the **MLEADER** command differ from the **QLEADER** command?
- 62. Define “content” of multiline leaders.
- 63. Label the parts of the multi-line leader illustrated below:



- 64. Describe how annotative text differs from normal text.
- 65. When would you use the **OBJECTSCALE** command?



Placing Dimensions

Dimensions play an important role in defining the size, location, angle, and other attributes of objects in drawings. Dimensions show the length and width of objects, the diameter of holes, and the angles of sloped parts.

This chapter describes how to place dimensions in the drawing using these commands:

DIMLINEAR places horizontal, vertical, and rotated dimensions.

DIMALIGNED places dimensions aligned with objects.

DIMBASELINE and **DIMCONTINUE** add baseline and continuous dimensions.

QDIM generates a variety of continuous dimensions.

DIMRADIUS and **DIMDIAMETER** place radial and diameter dimensions.

DIMJOGGED draws radial or linear dimensions with jogged leader or dimension lines.

DIMARC measures the lengths of arcs.

DIMCENTER places center marks on arcs and circles.

DIMANGULAR places angular dimensions.

NEW TO AUTOCAD 2010

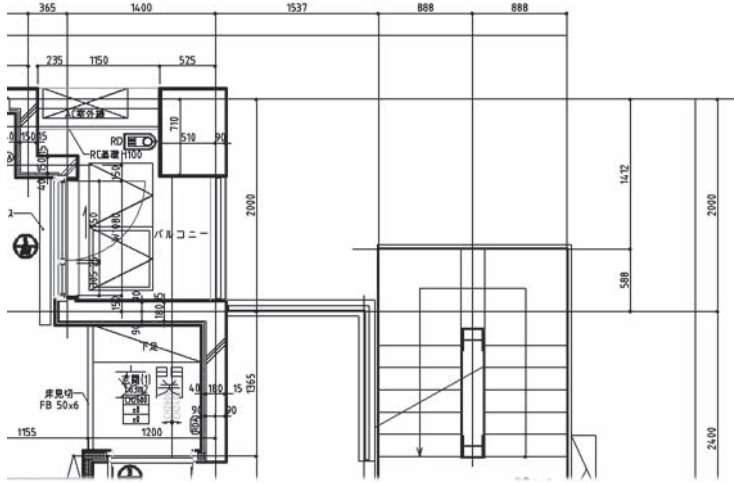
- The **DIMCONSTRAINT** command adds dimensional constraints to objects, as described separately in Chapter 13, “Applying Dimension Constraints.”

INTRODUCTION TO DIMENSIONS

Dimensions in drawings eliminate the need to use scale rulers and protractors to work out measurements and angles, and are far more accurate than measuring drawings with a ruler.

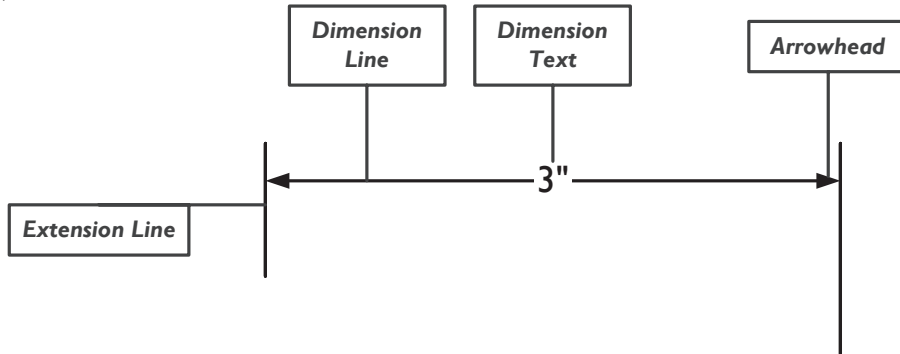
In some disciplines, such as architecture, the most important parts of the drawing are dimensioned, not every detail. In contrast, mechanical parts are completely dimensioned. Either way, dimensions should not interfere with the drawing.

The figure below illustrates metric dimensions in a sample Chinese drawing provided with AutoCAD.



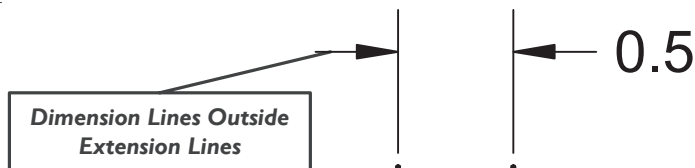
THE PARTS OF DIMENSIONS

Dimensions consist typically of these parts: a dimension line with an arrowhead at each end, a pair of extension lines, and dimension text indicating the measured distance.

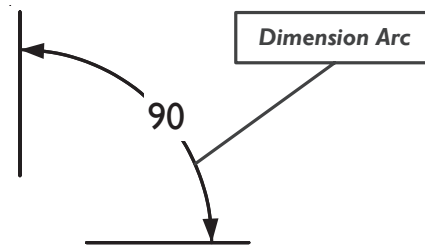


Dimension Lines

The *dimension line* indicates the distance or angle being measured. It is the line with the arrows or “ticks” at each end. When there is too little room between the extension lines, the dimension line is placed outside the extension lines or, in some cases, not drawn.



When you measure angles, the dimension line is an arc, instead of a straight line.



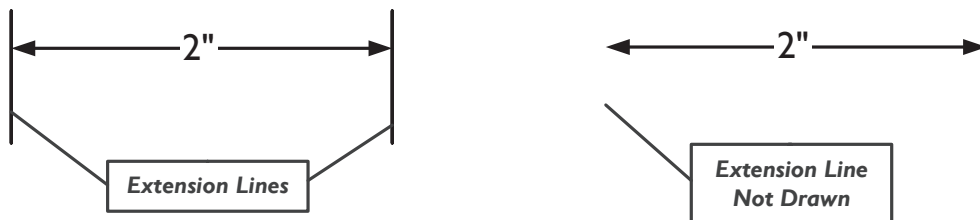
Dimension lines normally take on the properties of their layer. AutoCAD allows you to override the layer setting by specifying the lineweight, color, and linetype for the dimension and extension lines separately.

To report the distance or angle, text is placed on or near the dimension line. AutoCAD automatically creates a gap in the dimension line to make room for the text.

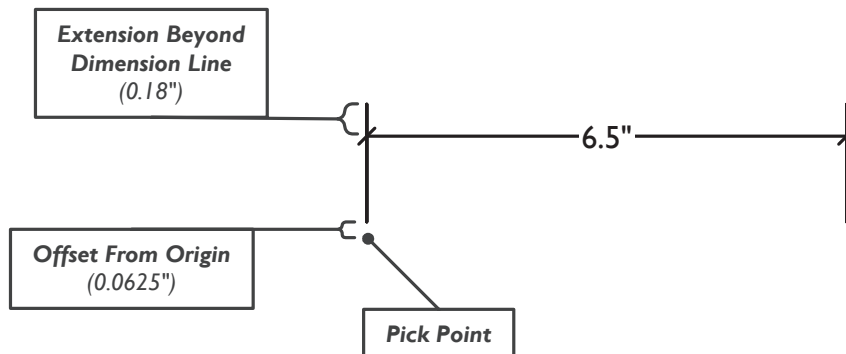
Extension Lines

Nearly all dimensioning commands begin by prompting you to pick two points: these become the start points for *extension lines*. Extension lines (sometimes called “witness lines”) indicate the geometric features being dimensioned. Extension lines are usually perpendicular to the dimension line, but need not be.

There are usually two extension lines, one at either end of the dimension line. Sometimes, the first or second extension line is not drawn, and in rare cases, neither is drawn.



AutoCAD *offsets* the extension lines from the pick points by a distance of 0.0625 units to mimic standard drafting practice. The offset distance is named “offset from origin,” and can be changed in the Dimension Style Manager dialog box.



In addition, AutoCAD draws the extension lines a short distance beyond the dimension line. Again, this is standard drafting practice. The default distance is 0.18 units, but can be changed in the Dimension Styles Manager dialog box.

AutoCAD normally draws the extension lines long enough to span the distance from the origin to just beyond the dimension line — but AutoCAD can also allow the length of extension lines to be fixed.

- Closed filled
- Closed blank
- ≡ Closed
- Dot
- ↗ Architectural tick
- ↗ Oblique
- ≡ Open
- ⊙ Origin indicator
- ⊙ Origin indicator 2
- ➔ Right angle
- ➔ Open 30
- ◆ Dot small
- Dot blank
- ◇ Dot small blank
- Box
- Box filled
- △ Datum triangle
- ▲ Datum triangle filled
- ↗ Integral
- None

Arrowheads

Arrowheads are placed at either end of the dimension line, pointing to the extension lines. If there is too little room between the extension lines, the arrowheads are placed outside. The Dimension Style Manager dialog box lets you specify whether arrowheads or text should first be moved outside, or forced to fit inside the extension lines.

The arrowheads can be replaced with tick marks and other symbols. AutoCAD includes many different arrowheads, and you can define your own. The figure illustrates the arrowhead styles included with AutoCAD; to create custom arrowheads, you first define them as blocks.



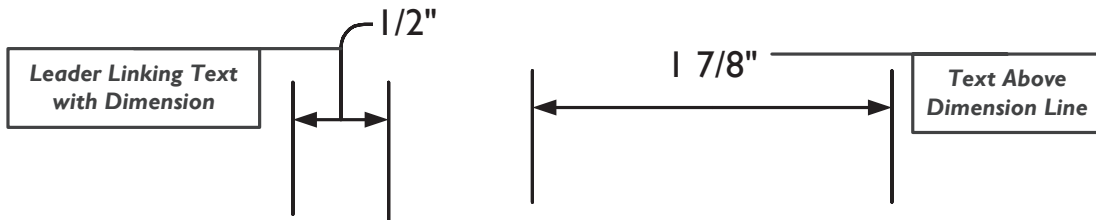
Arrowheads are usually drawn with a 1:3 aspect ratio: the arrow is three times longer than it is wide. Other arrowheads, such as tick marks and circles, are 1:1.

The length of the arrowhead varies, depending on the scale of the drawing. Arrowheads that are too small or large prove difficult to read and can distract. Generally, arrowheads are 1/8 inch in length when used in small drawings, and 3/16 inch in larger drawings.

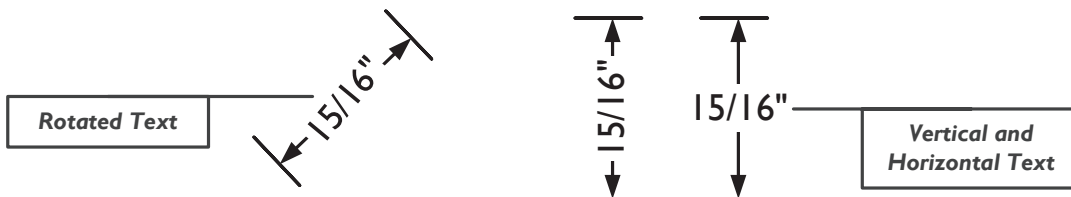
Dimension Text

The *dimension text* reports the distance or angle between the extension lines. Sometimes, the text reports other information, such as tolerances or alternate units of measurement. Typically you let AutoCAD measure the distance and determine the text; you can, however, override it and enter your own text with the mtext editor.

Dimension text appears in or near the dimension line, depending on whether there is enough room on the drafting standard. In architectural and structural drawings, the dimension is placed on top of the dimension line. In mechanical drawings, it is usually placed within the dimension line. When there is too little room between the extension lines, the text is placed outside, or even some distance away, and in some cases referenced with a leader line.



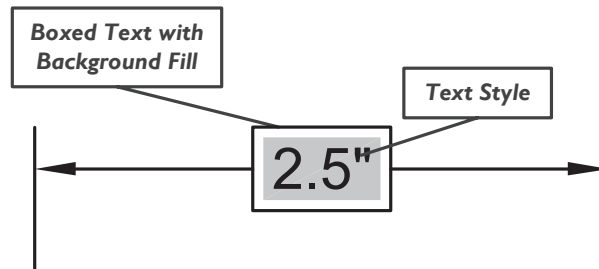
When the dimension text is horizontal and centered, the text is said by AutoCAD to be in the “home” position. Dimension text can be centered, left justified, or right justified on the dimension line. The text can be horizontal, vertical, or rotated.



Feet and inches are separated by a dash, as in 5'-4". If there are no inches, a zero is usually included: 6'-0". AutoCAD allows you to choose whether you want 0 feet and/or 0 inches displayed. If dimensions are stipulated strictly in inches, such as 72", use the inch mark to avoid confusion with feet or other units; in mechanical design, inch marks are not used.

Fractions are given either as common fractions, such as 1/2, 3/4, 9/16, and so on, or as decimal fractions, such as 0.50 and 0.75. In CAD, some text fonts lack “stacked” fractions, such as 1/2 and 3/4. For this reason, the dash is used to separate inches from fractions and to avoid confusion, such as 3-1/2".

Dimension text indirectly follows the text style set by the STYLE command and the numerical format and precision set by the UNITS command. The style, units, and precision of dimensions are determined by the DIMSTYLE command. You can specify the style name, color of text, fill (background) color, and fixed height, as well as decide whether to surround the text with a box.

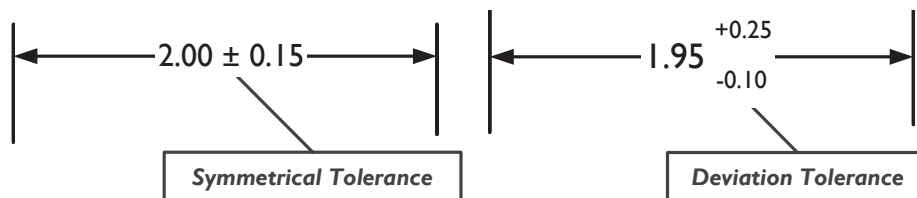


Annotative Scaling

Like text and leaders, dimension can be annotatively scaled. This ensures that they appear at the correct size at specific model space scale factors, and eliminates the need to calculate a scale factor to make dimensions plot correctly.

Tolerance Text

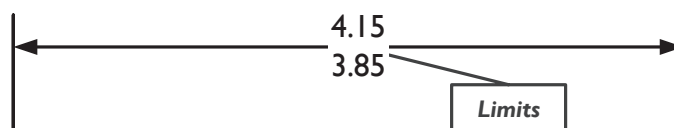
Dimension *tolerances* are plus and minus amounts appended to the dimension text. AutoCAD can add the tolerances automatically, but typically you specify the plus and minus amounts, which can be equal (“symmetrical”) or unequal (“deviation”).



If tolerances are symmetrical, they are drawn with the plus/minus symbol. If deviation, they are drawn one above the other: above for the plus amount, below for the minus.

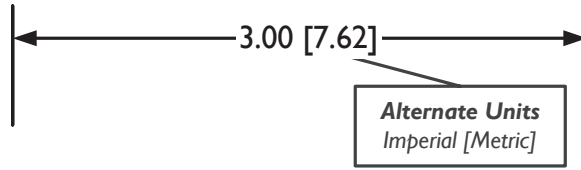
Limits Text

Instead of showing dimension tolerances, the tolerance can be applied to the text itself — added to and subtracted from the dimension text. This is called “limits.” The example below is a measurement of 4.00 units, with a tolerance of ± 0.15 .



Alternate Units

“Alternate units” show two forms of measurement, such as English and metric, on the same dimension line. The second set is usually shown in square brackets.



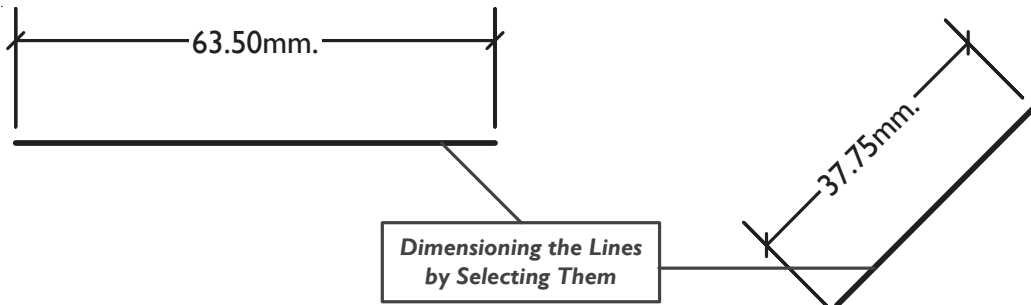
When alternate units are turned on in the dimension style, AutoCAD adds them automatically. When turned off, you can have AutoCAD calculate and add alternate units by entering a pair of square brackets ([]) when editing the dimension text.

DIMENSIONING OBJECTS

AutoCAD places dimensions between any two points in a drawing. In addition, AutoCAD has a *direct dimensioning* mode, where it dimensions specific objects automatically: lines, polyline segments, arcs, polyline arcs, circles, vertices, and single points.

Lines and Polyline Segments

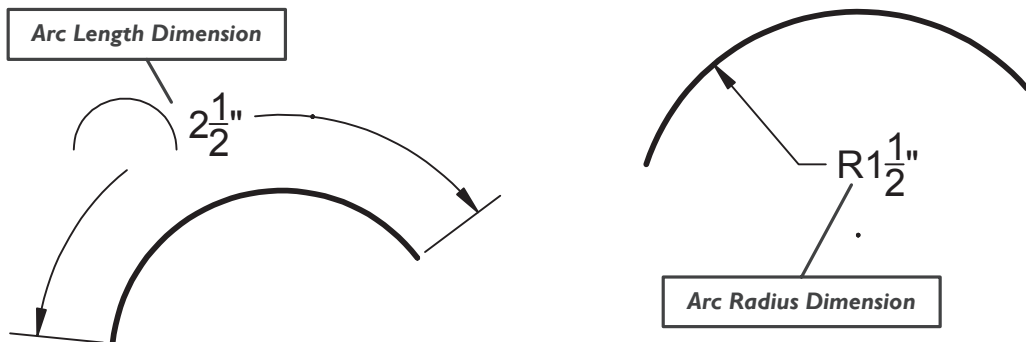
AutoCAD can dimension the lengths of lines and polyline segments with a single pick. The dimension is drawn completely, consisting of the dimension and extension lines, arrowheads, and text. This is accomplished typically with the DIMLINEAR and DIMALIGNED commands, as detailed later.



Arcs, Circles, and Polyarcs

Arcs are dimensioned by their radius or length. When dimensioning the radius, place the leader line at an angle to avoid horizontal and vertical placements. The dimension line for the length of an arc follows the curve of the arc. The arc symbol designates arc length dimensions.

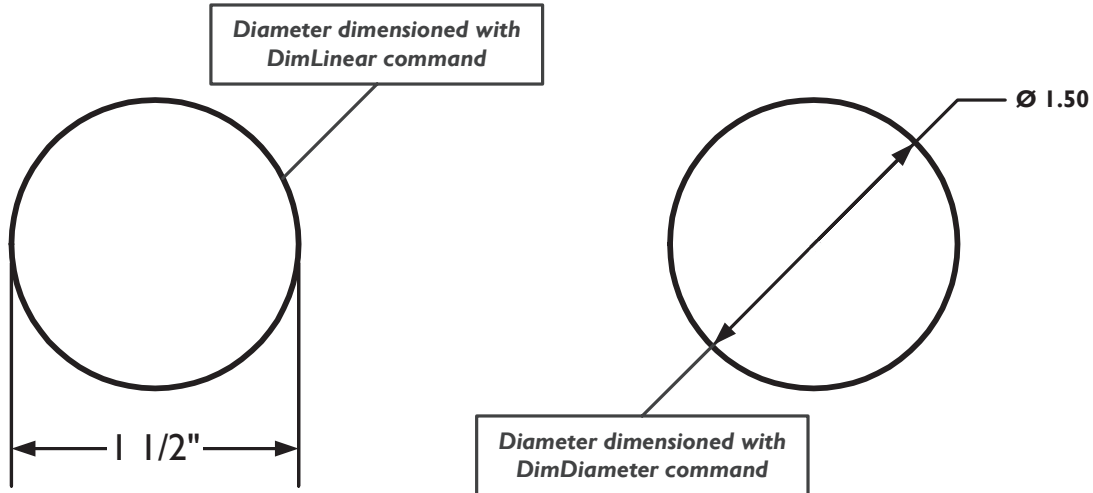
The letter “R,” designating radius, prefixes the dimension text, such as **R2.125**. If a circle is dimensioned by its diameter, the dimension text is prefixed by the diameter symbol, such as **Ø4.25**.



Objects constructed of several arcs are dimensioned in two stages: (1) locating the center of the arcs with horizontal or vertical dimensions, and (2) showing their radii with radius dimensioning.

AutoCAD dimension arcs and polyline arcs with a single pick. The resulting dimension depends on the command you use:

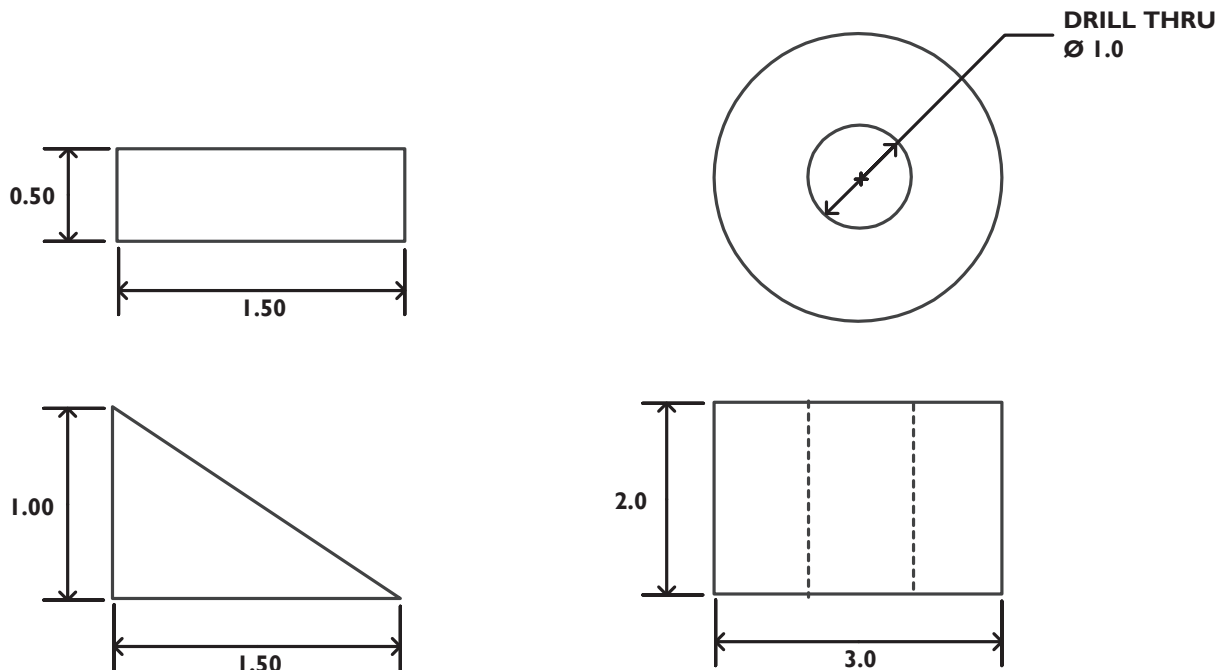
The DIMLINEAR and DIMALIGNED commands place dimension and extension lines, arrowheads, and text. The DIMRADIUS and DIMDIAMETER commands place dimension lines or leaders, arrowheads, and text.



Arcs are usually dimensioned with radii, while circles are usually dimensioned with diameters.

Wedges and Cylinders

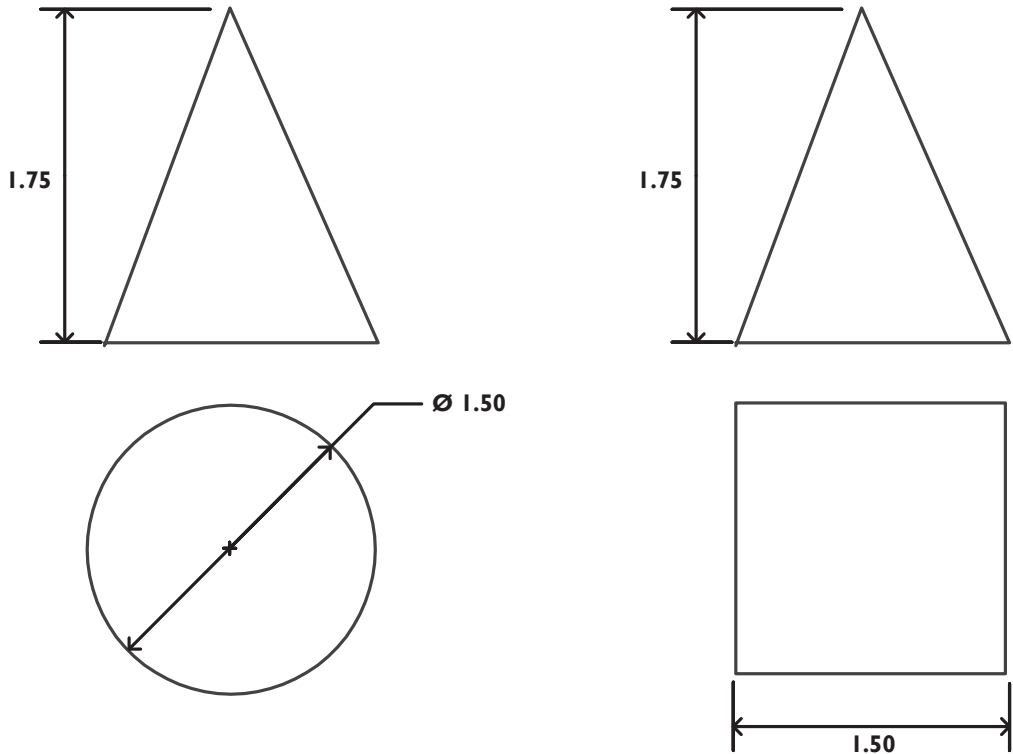
Wedges are dimensioned in two views using three distances: length, width, and height.



Cylinders are dimensioned for diameter and height. The diameter is typically dimensioned in the circular view. If a drill-through is dimensioned, it is described by a diameter leader.

Cones and Pyramids

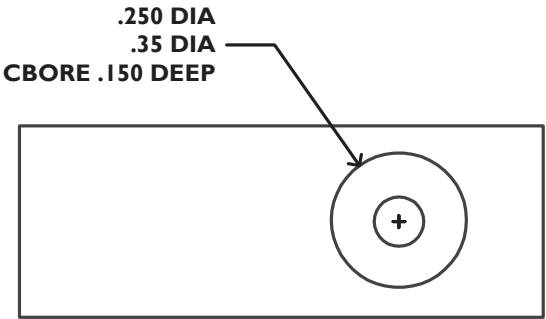
Cones are dimensioned at the diameter and the height. Some conical shapes, such as truncated cones, require two diameter dimensions. Pyramids are dimensioned like cones.



Holes

Holes are created by drilling, reaming, boring, punching, or coring materials. You should dimension holes in drawings with notes that give the diameter, operation, and the number of holes, if more than one. The operation describes such types of holes as counter-bored, reamed, and countersunk.

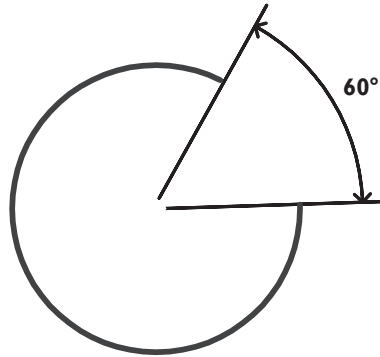
Whenever possible, point the dimension leader to the hole in the circular view. Holes made up of several diameters can be dimensioned in their section.



Standards dictate that drill sizes be designated as decimal fractions.

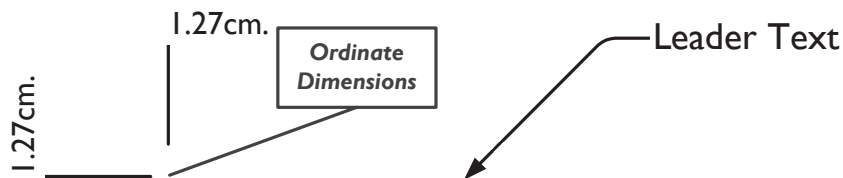
Vertices

AutoCAD measures the angle of lines, arcs, and polyline arcs with two picks using the DIMANGULAR command.



Single Points

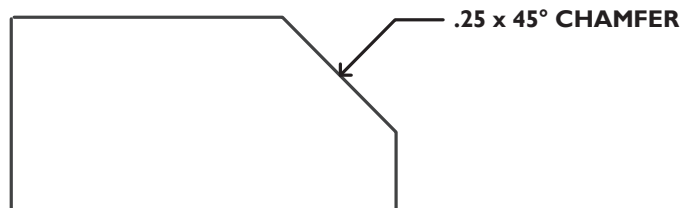
To “dimension” a single point, you typically use the LEADER, QLEADER, MLEADER, or DIMORDINATE commands. Leaders are typically pieces of text that point at a spot in the drawing through lines. Ordinate dimensions measure the x and y coordinates of objects relative to a base point.



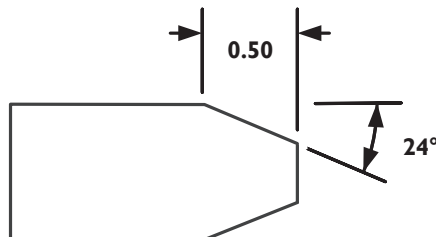
For drawing leaders, see Chapter 10, “Placing and Editing Text.”

Chamfers and Tapers

A chamfer is an angled surface applied to an edge. Use a leader to dimension chamfers of 45 degrees, with the leader text designating the angle and one (or two) linear distances.

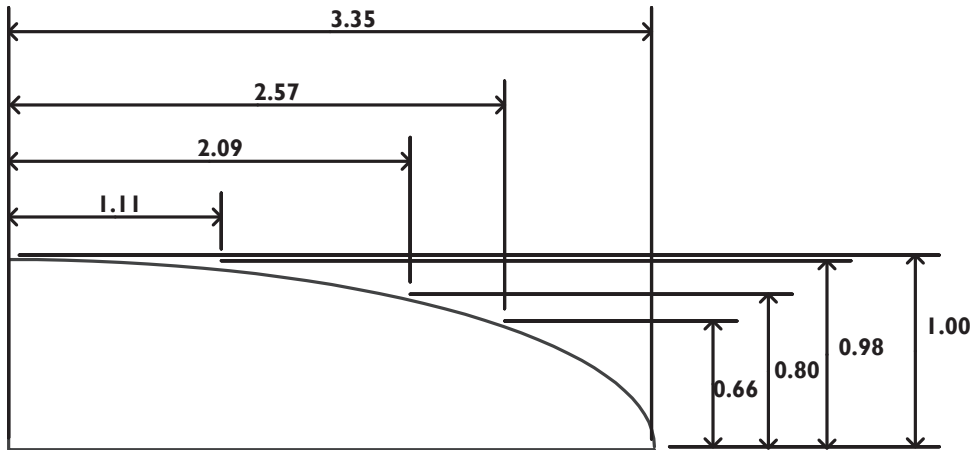


If the chamfer is not 45 degrees, dimensions showing the angle and the linear distances describe the part.



A taper can be described as the surface of a *cone frustum* (cone with the top sliced off). Dimension

tapers by giving any three of these parameters: start diameter, end diameter, rate of taper, and/or the length.



DIMENSION MODES

AutoCAD constructs dimensioning *semiautomatically*. All it needs from you is some basic information, such as what to dimension, or where to start and end the dimensions. Then it constructs the dimension for you, measures the distances and angles, and then draws all the elements of the dimension.

AutoCAD's dimensions are *associative*. That means they automatically update when associated objects are moved and stretched. This is a very powerful feature and a great time-saver.

Dimensions, however, do not need to be associative; AutoCAD can work with several levels of associativity:

Fully associative is the standard mode, where AutoCAD attaches the ends of the extension lines to the geometry of the object. Move the object, and the dimension moves with it; stretch the object, and the dimension text updates automatically. This is the preferred method.

Partly associative was Autodesk's first attempt at associativity. Extension lines are attached to *dimension points* placed on layer Defpoints (short for "definition points"). When stretching or moving objects, you must include the defpoints in the selection set; otherwise, the dimension does not update correctly.

Non-associative means the dimension is not attached to objects in any way.

Trans-spatial means that dimensions created in paper space are attached to objects in model space; changing the model updates the dimensions in paper space.

Dimensional constraints define distances within and between objects parametrically. In this case, the dimension drives the distance, either with a value (number) or a formula parameter ([new to AutoCAD 2010](#)). Dimensional constraints are covered in Chapter 13, "Applying Dimensional Constraints."

Dimensions are placed as if they were blocks: the lines, arrows, and text act as a single object. You can use the EXPLODE command to break apart dimensions.



Note When AutoCAD reports "Non-associative dimension created," this means that it was not able to attach a dimension to the object.

Dimension Variables and Styles

Dimension variables determine how the dimensions are drawn; they are system variables specific to dimensions, sometimes called “dimvars” for short. Some dimvars store values, such as the color of the dimension line and the look of the arrowhead; others are toggles that turn values on and off, such as whether the first or second extension line should be displayed.

Just as styles determine the look of text, *dimension styles* determine the look of dimensions. Dimension styles (“dimstyles” for short) are created and modified with the **DIMSTYLE** command, which collects all the settings of dimension variables. Dimstyles are discussed in Chapter 12, “Editing Dimensions,” and dimvars are listed in Appendix D, “Summary of Dimensional Variables.”

Dimension Standards

There are many ways to draw dimensions, because different industries and countries define standards differently. AutoCAD includes the ISO-25 dimensioning standard with its *acadiso.dwt* template drawing. (Other standards which were included in previous releases of AutoCAD, such as DIN, JIS, and Gb, are no longer part of AutoCAD.)

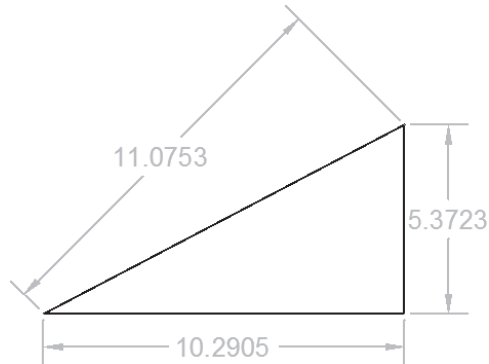
Most standards organizations make the dimension standards available to members at a cost (for example at www.buildingsmartalliance.org/ncs, — that’s .org, not .com), while some are freely available at Web sites (for example at www.cadinfo.net/editorial/archdim1.htm).

Object Snaps

Object snap modes are very helpful in placing dimensions accurately. For example, the **END**point and **INT**ersection osnaps are useful for capturing the ends and intersections of lines, while the **CEN**ter and **TAN**gent osnaps help with dimensions of arcs and circles.

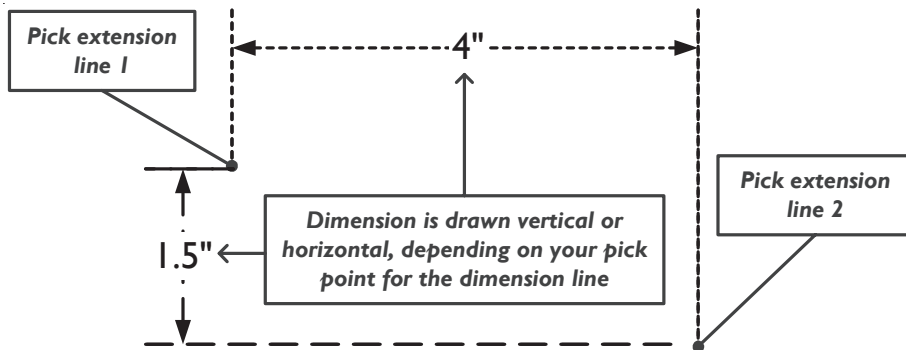
DIMLINEAR

The DIMLINEAR command places horizontal, vertical, and rotated dimensions.



*Examples of rotated, horizontal, and vertical dimensions.
The rotated dimension was placed at 45 degrees.*

You specify the location of the extension lines, and AutoCAD determines whether to draw a horizontal or vertical dimension automatically — based on where you place the dimension line.

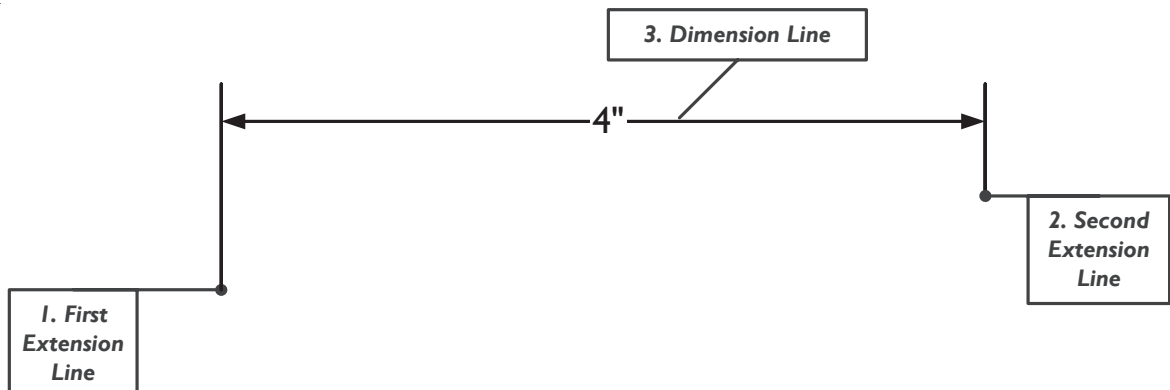


This command also applies dimensioning directly to objects.

TUTORIAL: PLACING DIMENSIONS HORIZONTALLY

The command places horizontal dimensions. (If you find it difficult to draw dimensions horizontally, you can use the undocumented DIMHORIZONTAL command, which draws only horizontal dimensions.)

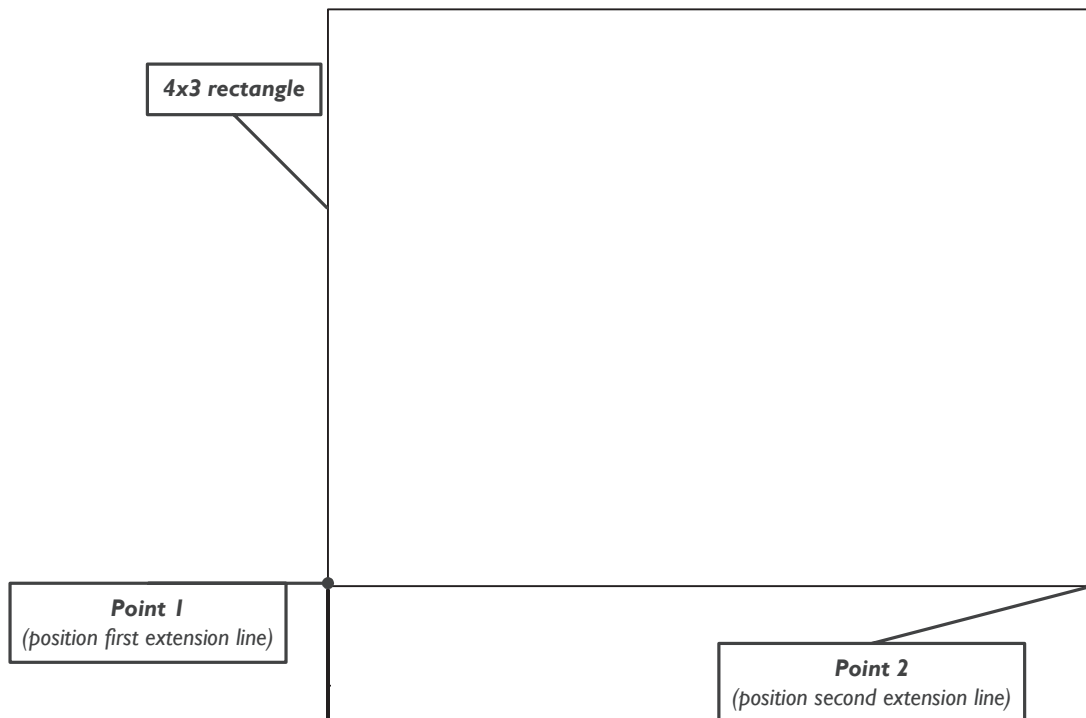
To locate extension lines, you select an object (line, arc, circle, or polyline) or pick two points (1 and 2, in the figure below). Another pick point (3) is required to locate the dimension line. The text and arrowheads are located automatically by AutoCAD.



In this tutorial, you dimension a rectangle.

1. Start AutoCAD with a new drawing using the *acad.dwt* template. Ensure that **ENDpoint** object snap is turned on.
2. Draw a 4x3 rectangle with the **RECTANGLE** command.
3. To dimension the width of the rectangle (horizontally), start the **DIMLINEAR** command:
 - In the ribbon's Annotation tab, choose the **Linear** button in the Dimensions panel.
 - At the 'Command:' prompt, enter the **DIMLINEAR** command.
Command: **dimlinear** (Press ENTER.)
 - Alternatively, enter the aliases **dli** or **dimlin** at the 'Command:' prompt.
4. At the prompt, pick the lower left corner of the rectangle:
Specify first extension line origin or <select object>: (Pick point 1 to locate first extension line.)

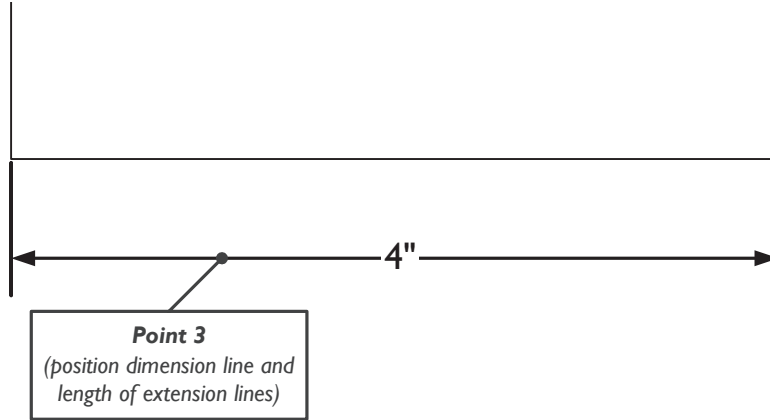
This point specifies the origin of the first extension line.



5. Pick the other lower corner of the rectangle:
Specify second extension line origin: (Pick point 2 to locate second extension line.)
Point #2 specifies the origin of the second extension line, and measures the distance reported by the dimension.

6. Pick a third point to located the dimension line:

Specify dimension line location or [Mtext/Text/Angle]: *(Pick point 3 to place the dimension line.)*



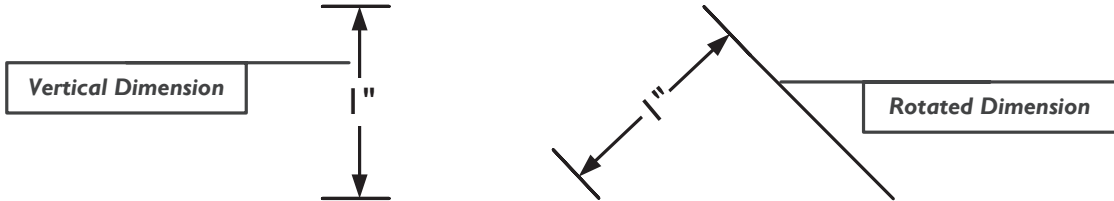
Point #3 specifies the distance of the dimension line away from the object; this point also determines the length of the two extension lines.

Notice that AutoCAD reports the length measurement at the command prompt, and draws the horizontal dimension.

Dimension text = 4"

TUTORIAL: PLACING DIMENSIONS VERTICALLY AND ROTATED

The DIMLINEAR command also places dimensions vertically and at an angle (rotated).



To place dimensions vertically, pick the two points so that they are roughly vertical.

To place a rotated dimension, enter the DIMLINEAR command and then choose the **Rotated** option before locating the dimension line, as follows:

Command: **dimlinear**

Specify first extension line origin or <select object>: *(Pick a point.)*

Specify second extension line origin: *(Pick another point.)*

Type **r** to specify the **Rotated** option:

Specify dimension line location or

[Mtext/Text/Angle/Horizontal/Vertical/Rotated]: **r**

AutoCAD asks you for the angle:

Specify angle of dimension line <0>: **45**

And then place the dimension line:

Specify dimension line location or

[Mtext/Text/Angle/Horizontal/Vertical/Rotated]: *(Pick a point.)*

If you find the process frustrating, you can use the DIMVERTICAL and DIMROTATED commands to force AutoCAD to place the dimensions correctly.

Dimensioning Objects

The **DIMLINEAR** command lets you dimension objects directly. This means that you don't need to pick points for the first and second extension line; AutoCAD finds the points for you.

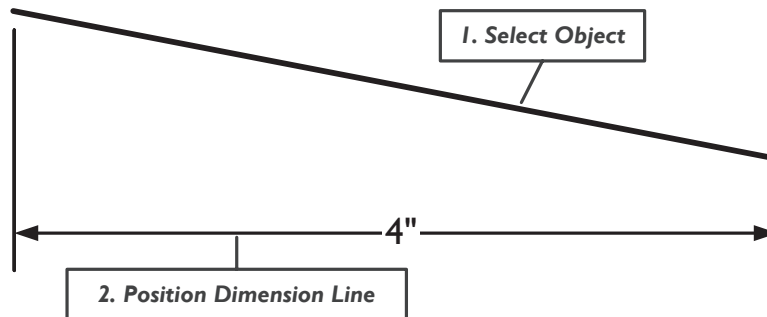
This allows speedy, repeated dimensioning of drawings:

- a. Press **SPACEBAR** to repeat the command.
- b. Press **SPACEBAR** to choose the **Select Object** option.
- c. Pick the object.
- d. Pick the location of the dimension line.
- e. Repeat: **SPACEBAR** – **SPACEBAR** – pick – pick.

TUTORIAL: DIMENSIONING OBJECTS

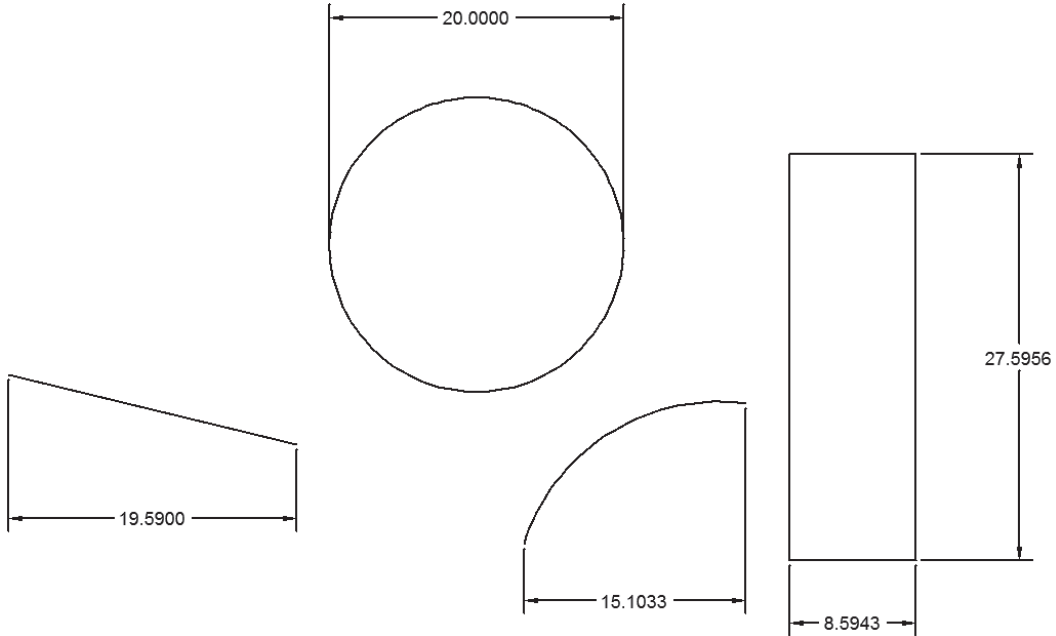
In this tutorial, you use the **dimlinear** command's **Select Object** option to dimension a variety of geometry directly.

1. From the Companion CD, open the *Dimension Objects.dwg* file, a drawing of several geometric shapes.
2. Start the **DIMLINEAR** command, and then press **ENTER** to dimension objects, as follows:
Command: **dimlinear** (Press **ENTER**.)
Specify first extension line origin or <select object>: (Press **Enter**.)
3. AutoCAD prompts you to pick an object. Choose the line:
Select object to dimension: (Pick the line.)



4. AutoCAD asks you to pick a location for the dimension line:
Specify dimension line location or
[Mtext/Text/Angle/Horizontal/Vertical/Rotated]: (Position the dimension line.)
And then AutoCAD reports the length of the line in the command bar:
Dimension text = 4.000
5. Continue the tutorial by dimensioning the circle:
 - a. Press **ENTER** to repeat the **DIMLINEAR** command:
Command: (Press **ENTER**.)
 - b. Press **ENTER** to select an object:
Specify first extension line origin or <select object>: (Press **Enter**.)
 - c. Pick the circle:
Select object to dimension: (Pick the circle.)
 - d. Locate the dimension line:
Specify dimension line location or
[Mtext/Text/Angle/Horizontal/Vertical/Rotated]: (Position the dimension line.)

6. Repeat to dimension the arc and the sides of the rectangle.



LINEAR DIMENSIONS: ADDITIONAL METHODS

The DIMHORIZONTAL command has options to change the dimension text.

- **MText** displays the mtext editor.
- **Text** prompts you to edit the dimension text.
- **Angle** rotates the dimension text.
- **Horizontal** forces the dimension line horizontally.
- **Vertical** forces the dimension line vertically.
- **Rotated** rotates the dimension line.
- **Angle** rotates the dimension text.

Let's look at each option.

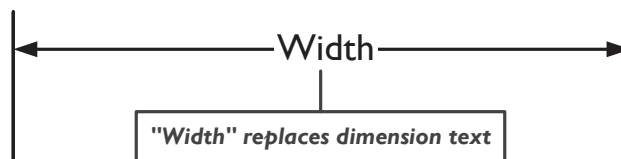
MText

The **MText** option displays the mtext editor, allowing you to edit the dimension text. The editor should be familiar to you from Chapter 10, "Placing and Editing Text."

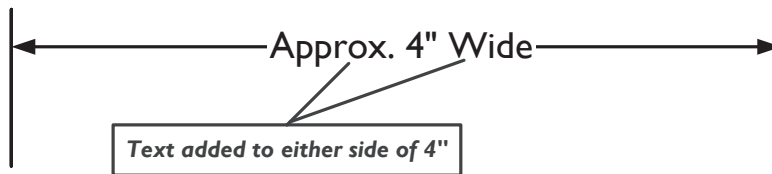
The mtext editor highlights the default dimension text. (Prior to AutoCAD 2006, the double angle bracket <> was used as a shorthand notation for the default dimension text.)

Here are some examples of how you can work with the default text:

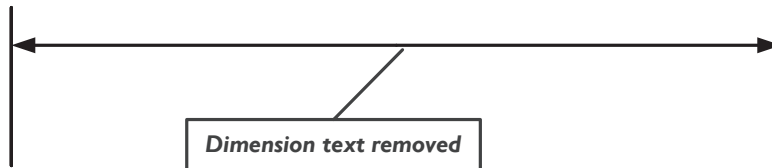
- Replace default text with other text, and AutoCAD shows the new text on the dimension line.



- Add text on either side of the default text, and AutoCAD shows the added text.



- Replace the default text with a space character, and AutoCAD erases the dimension text.



Special Symbols

To add special characters, use the mtext editor's **Symbols** submenu.

Alternatively, you can enter metacharacters to show these common symbols:

- ° (degrees) with %%d
- ± (plus or minus) with %%p
- ∅ (diameter) with %%c

Text

The **Text** option prompts you to edit the dimension text at the command prompt.

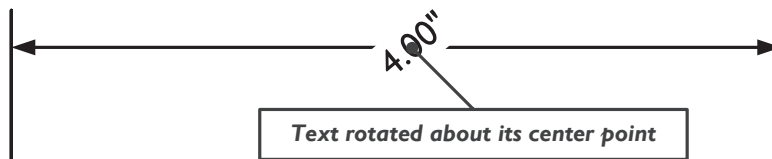
Enter dimension text <4.0000>: **The distance is <> inches**

Include the double angle brackets (<>) to specify the location of AutoCAD's measured distance with the text you add. The result is: **The distance is 4.0000 inches.**

Angle

The **Angle** option rotates the dimension text about its center point.

Specify angle of dimension text: *(Enter an angle, such as 45, or use the mouse to show the angle.)*



Horizontal, Vertical and Rotated

The **Horizontal** option forces the dimension line to the horizontal, no matter where you locate the dimension line. This makes it operate like DIMHORIZONTAL.

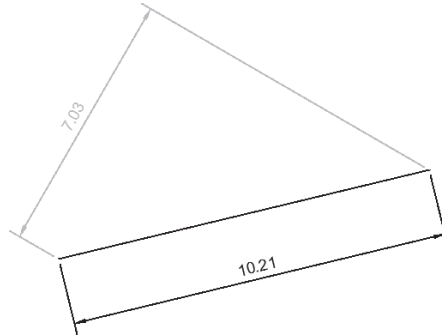
Similarly, the **Vertical** option forces the dimension line to the vertical, just like the DIMVERTICAL command.

The **Rotated** option prompts you for an angle, and then draws the dimension line at that angle (measured from the positive x-axis), adjusting the lengths of the extension lines as required.

DIMALIGNED

The DIMALIGNED command places dimensions aligned to two points.

This command is like the DIMLINEAR command, except that the pick points for the extension lines determine the angle of the dimension line. DIMALIGNED reports the true lengths of objects.



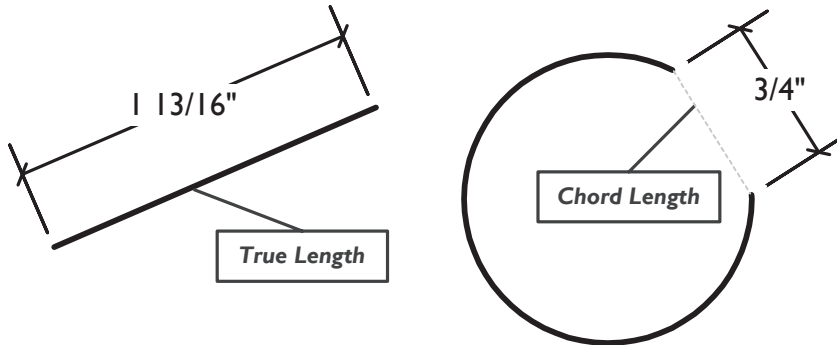
In gray: Rotated dimension.

In black: Aligned dimension.

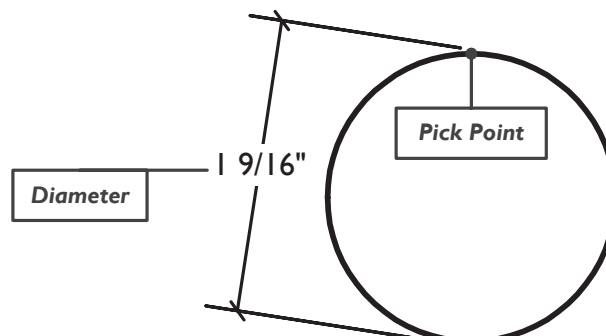
(Don't confuse *rotated* dimensions with *aligned* ones. In a rotated dimension, you specify the angle of the dimension line, distorting the lengths of the extension lines; in aligned dimensions, AutoCAD aligns the dimension line with the extension lines or the object being dimensioned. For the figure above, I specified an angle of 60 degrees for the rotated dimension.)

When you employ direct dimensioning, press ENTER and then select an object; DIMALIGNED reacts differently, depending on the object you select:

Lines, Arcs, and Polylines — endpoints of lines, arcs, and polylines determine the angle of the dimension line; AutoCAD measures the length of the line and polyline segments and the chord length of arcs.



Circles — a point on the circle determines the angle of the dimension line; AutoCAD measures the circle's diameter.



PLACING ALIGNED DIMENSIONS

1. In AutoCAD, draw a line at an angle.
2. To place a dimension aligned to the line's angle, start the **DIMALIGNED** command:
 - In the ribbon's Annotation tab, choose the **Aligned** button in the Dimensions panel.
 - At the 'Command:' prompt, enter the **dimaligned** command.
Command: **dimaligned** (Press ENTER.)
 - Alternatively, enter the aliases **dal** or **dimali** at the 'Command:' prompt.
3. In all cases, AutoCAD prompts you to select the object you want dimensioned:
Specify first extension line origin or <select object>: (Press ENTER.)
Select object to dimension: (Pick one object.)
4. Pick a point to place the dimension line:
Specify dimension line location or
[Mtext/Text/Angle]: (Pick a point to place the dimension line.)
Dimension text = 10.0

This command's **Extension line**, **Mtext**, **Text**, and **Angle** options operate identically to those of the **DIMLINEAR** command.

DIMBASELINE AND DIMCONTINUE

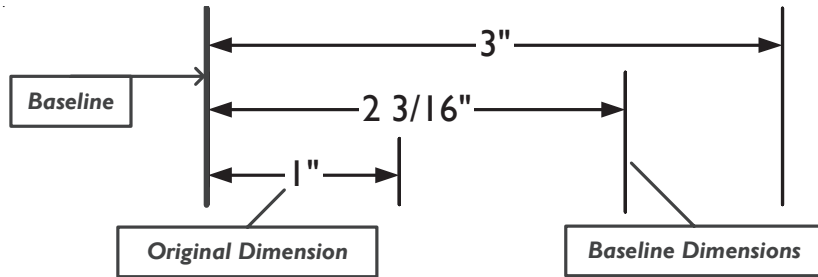
The **DIMBASELINE** command (also called “parallel dimensioning”) continues dimensions from baselines, while the **DIMCONTINUE** command (also called “chain dimensioning”) continues dimensions from the previous dimension.

Baseline dimensions tend to stack over one another, while *continuous* dimension are usually located next to each other. The stack distance is 0.38, but can be changed. Continuous dimensions share common extension lines.

Both commands continue from the last placed dimension, which can be linear, angular, or ordinate. (These commands do not work with leaders, radial, or diameter dimensions.) You cannot use either command until at least one other dimension has been placed in the drawing. When no dimensions were created during the current drafting session, AutoCAD prompts you to select a dimension as the base.

TUTORIAL: PLACING ADDITIONAL DIMENSIONS FROM A BASELINE

1. To place baseline dimensions, start the **DIMBASELINE** command:
 - In the ribbon's Annotation tab, choose the **Baseline** button in the Dimensions panel.
 - At the 'Command:' prompt, enter the **dimbaseline** command.
Command: **dimbaseline** (Press ENTER.)
 - Alternatively, enter the aliases **dba** or **dimbase** at the 'Command:' prompt.
2. If necessary, AutoCAD prompts you to select a dimension to use as the base:
Select base dimension: (Pick a linear, angular, or ordinate dimension.)
3. AutoCAD prompts you to pick the endpoint for the next dimension:
Specify a second extension line origin or [Undo/Select] <Select>: (Pick a point.)
Dimension text = 10.0



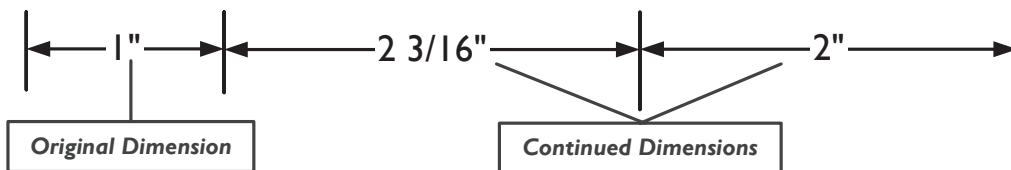
The baseline dimension continues from the first extension line of the previous dimension.

4. AutoCAD repeats the prompt for the next dimension:
Specify a second extension line origin or [Undo/Select] <Select>: *(Pick another point.)*
5. Press **ESC** to exit the command:
Specify a second extension line origin or [Undo/Select] <Select>: *(Press esc.)*

TUTORIAL: CONTINUING DIMENSIONS

The **DIMCONTINUE** command is identical to **DIMBASELINE**, except that it continues dimensions by placing new ones next to each other.

1. To continue dimensioning, start the **DIMCONTINUE** command:
 - In the ribbon's Annotation tab, choose the **Continue** button in the Dimensions panel.
 - At the 'Command:' prompt, enter the **dimcontinue** command.
Command: **dimcontinue** *(Press ENTER.)*
 - Alternatively, enter the aliases **dco** or **dimcont** at the 'Command:' prompt.
2. If necessary, AutoCAD prompts you to select a dimension to use as the base:
Select base dimension: *(Pick a linear, angular, or ordinate dimension.)*
3. AutoCAD prompts you to pick the endpoint for the next dimension:
Specify a second extension line origin or [Undo/Select] <Select>: *(Pick a point.)*
Dimension text = 10.0



Notice that the dimension continues from the second extension line of the previous dimension.

4. AutoCAD repeats the prompt for the next dimension:
Specify a second extension line origin or [Undo/Select] <Select>: *(Pick another point.)*
5. Press **ESC** to exit the command:
Specify a second extension line origin or [Undo/Select] <Select>: *(Press esc.)*

The **Select** option selects another dimension from which to continue. AutoCAD prompts:

Select continued dimension: *(Pick an extension line.)*

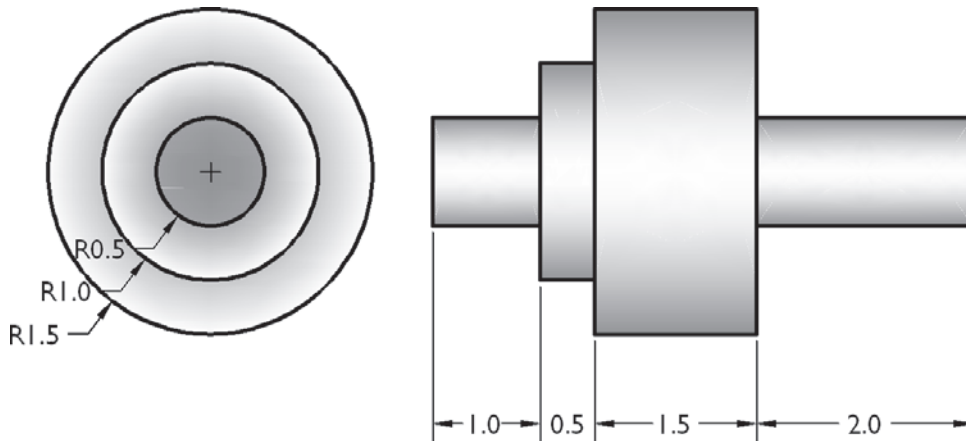
By picking an extension line, you determine the direction in which the dimension continues — to the left or to the right — assuming the dimension is horizontal.



The **QDIM** command places baseline and continuous style dimensions in one fell swoop.

It is a more powerful version of the **DIMCONTINUE** and **DIMBASELINE** commands: it places continued radial dimensions, applies the continued dimensions all at once, and more. Most importantly, it can dimension more than one object at a time. On the negative side, it is more complex than the two previous commands.

The figure below illustrates the continuous radius and continuous linear dimensions generated by this command. “While this is how AutoCAD dimensions radii,” notes the technical editor, “it is poor practice. Dimensions should always be *outside* the parts.”



Perhaps the most powerful aspect of **QDIM** is its ability to change the type of dimension. For instance, you might first dimension objects in staggered mode, but some time later want to change to ordinate mode. Making the change is this simple: restart the command, select the same objects, and then specify ordinate mode. AutoCAD changes the staggered dimensions to ordinate dimensions.

TUTORIAL: QUICK CONTINUOUS DIMENSIONS

1. To place continuous-style dimensions quickly, start the **QDIM** command:
 - In the ribbon's Annotation tab, choose the **Quick Dimension** button in the Dimensions panel.
 - At the 'Command:' prompt, enter the **qdim** command.

Command: **qdim** (Press ENTER.)

2. In all cases, AutoCAD reports the dimensioning priority, and then prompts you to select the geometry to dimension:

Associative dimension priority = Endpoint

Select geometry to dimension: (Select one or more objects.)

Select geometry to dimension: (Press ENTER to end object selection.)

Unlike other dimensioning commands, you may select more than one object.

3. Pick a point to define the position of the dimension line:

Specify dimension line position, or

[Continuous/Staggered/Baseline/Ordinate/Radius/Diameter/datumPoint/Edit/settings]
 <Radius>: (Pick a point.)

Depending on the objects you select, AutoCAD guesses at how to dimension them.

QUICK DIMENSIONS: ADDITIONAL METHODS

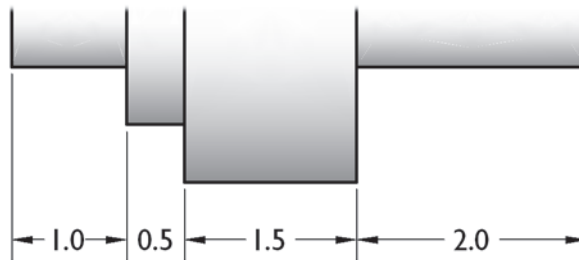
The QDIM command boasts a host of options to force the type of continuous dimension and specify settings:

- **Continuous** forces continued dimensions.
- **Staggered** places staggered dimensions.
- **Baseline** places baseline dimensions.
- **Ordinate** places ordinate dimensions.
- **Radius** places radial dimensions.
- **Diameter** places diameter dimensions.
- **datumPoint** specifies the start point for baseline and ordinate dimensions.
- **Edit** edits continuous dimensions.
- **seTtings** selects the object snap priority.

Let's look at all the options.

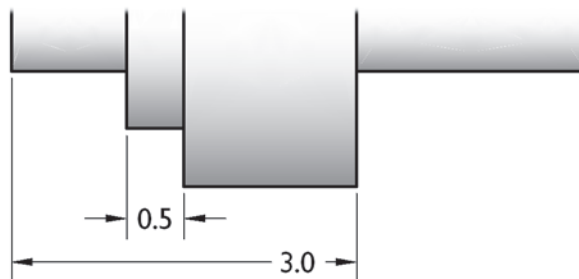
Continuous

The **Continuous** option forces continued dimensions, like those created by DIMCONTINUE.



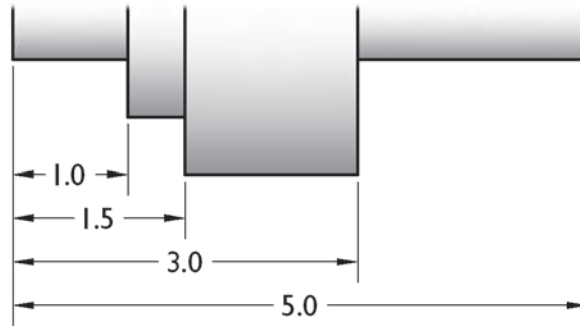
Staggered

The **Staggered** option forces staggered dimensions, which are like stacked continuous dimensions. This option, however, requires an even number of dimensionable objects. As illustrated in the figure, the odd-numbered segment, unfortunately, is left out and is not dimensioned.



Baseline

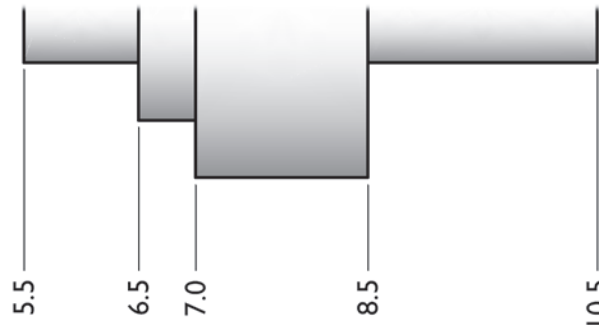
The **Baseline** option forces baseline dimensions, like those created by the DIMBASELINE command. The figure below shows the dimensions starting at the left end of the flywheel; you can change the start point with the **Datum point** option, as described later.



Ordinate

The **Ordinate** option forces ordinate dimensions, like those created by the **DIMORDINATE** command, with two differences. **DIMORDINATE** prevents ordinate dimensions from ending on top of each other, while **QDIM** does not. Also, **DIMORDINATE** measures relative to the UCS, while this command measures relative to a *datum point*.

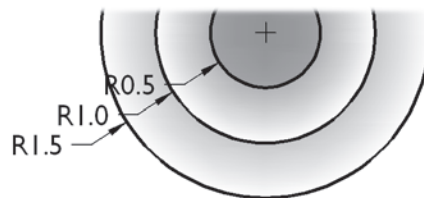
The figure illustrates x-ordinate dimensions starting with a value of 5.0, which is the distance from the origin (0,0). You can change the origin point with the **Datum point** option.



To create y-ordinate dimensions, move the cursor to the left or right side of the object during the “Specify dimension line position” prompt.

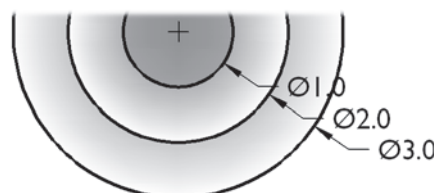
Radius

The **Radius** option forces radius dimensions, and works only with circles and arcs. Selecting other circular objects causes AutoCAD to complain, “No arcs or circles are currently selected.” The radial dimensions are placed at the point you pick at the “Specify dimension line position” prompt.



Diameter

The **Diameter** option forces diameter dimensions.



It operates identically to the **Radius** option. In both cases, it's a good idea to avoid placing dimensions horizontally, because they tend to bunch up. Because placing dimensions on parts is poor practice, you should manually move the radial and diameter dimensions. This is easily done with grips editing.

datumPoint

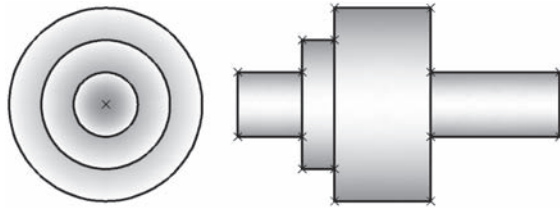
The **datumPoint** option prompts you to select a new base point from which to measure baseline and ordinate dimensions.

Select new datum point: *(Pick a point.)*

Edit

The **Edit** option enters editing mode, and prompts you to add and remove dimension markers. These markers show up as an **x** at the endpoints of lines, and the center points of circles and arcs.

Indicate dimension point to remove, or [Add/eXit] <eXit>: *(Enter an option.)*



Remove removes markers from objects. To remove, pick a marker with the cursor. AutoCAD reports, "One dimension point removed."

Add adds markers to the drawing. To add, pick a point anywhere in the drawing; it need not be on an object. AutoCAD reports, "One dimension point added."

eXit exits **Edit** mode, and returns to the "Specify dimension line position" prompt.

seTtings

The **seTtings** options doesn't set much, except whether endpoints or intersections have priority, and even then I am not convinced either setting makes any difference. AutoCAD displays the following prompt:

Associative dimension priority [Endpoint/Intersection]: *(Enter E or I.)*

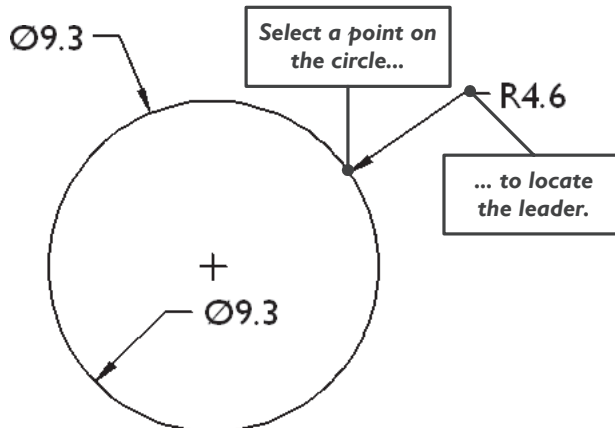


DIMRADIUS AND



DIMDIAMETER

The **DIMRADIUS** command places dimensions that measure the radius of circular objects, while the **DIMDIAMETER** command places dimensions that measure diameters.

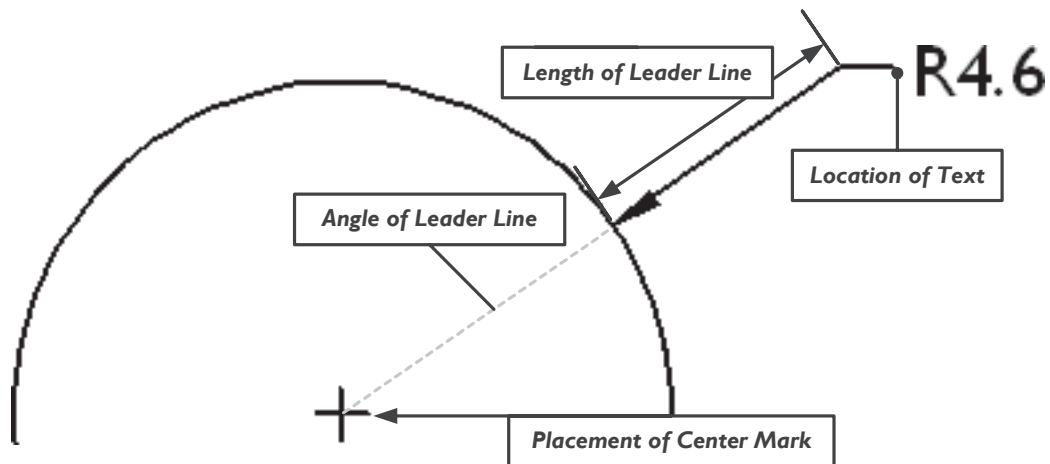


These commands dimension circles, arcs, and *polyarcs* (arcs that are parts of polylines). They do not dimension other curved objects, such as ellipses, splines, and regions.

The dimensions created by these two commands are more like leaders than the two-extension-line dimensions you have seen so far in this chapter. They don't prompt for a point, but immediately ask you to select a circle or an arc.

The second prompt asks you to "Specify dimension line location." Although this sounds like a single item (the dimension line), the point you pick determines several things at once, which can be difficult for new users:

- The angle of the leader line.
- The length of the leader line.
- The location of the dimension text.
- The placement of the center mark.
- The extent of an extension line.



TUTORIAL: PLACING RADIAL DIMENSIONS

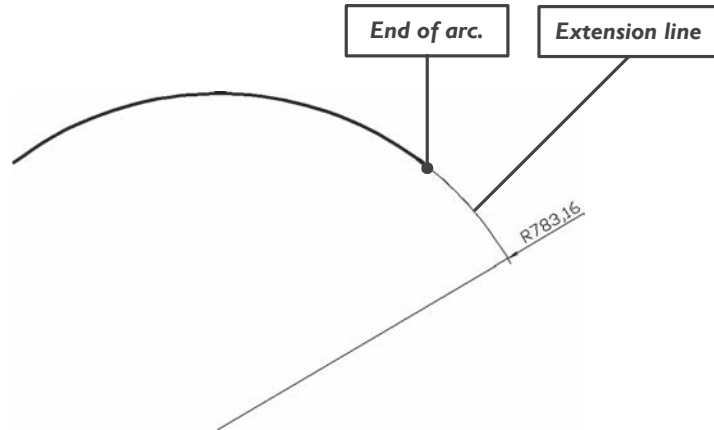
1. To dimension a radius, start the **DIMRADIUS** command:
 - In the ribbon's Annotation tab, choose the **Radius** button in the Dimensions panel.
 - At the 'Command:' prompt, enter the **dimradius** command.
Command: **dimradius** (Press ENTER.)
 - Alternatively, enter the aliases **dra** or **dimrad** at the 'Command:' prompt.
2. In all cases, AutoCAD prompts you to select the arc or circle you want dimensioned:

Select arc or circle: *(Pick one arc, circle, or polyarc.)*

Dimension text = 10.0
3. Pick a point to place the dimension leader:

Specify dimension line location or [Mtext/Text/Angle]: *(Pick a point to position the dimension leader.)*

When you position the dimension beyond the end of the arc, AutoCAD adds an extension line automatically, as illustrated below:



The DIMDIAMETER command operates identically:

Command: **dimdiameter**

Select arc or circle: *(Pick one arc, circle, or polyarc.)*

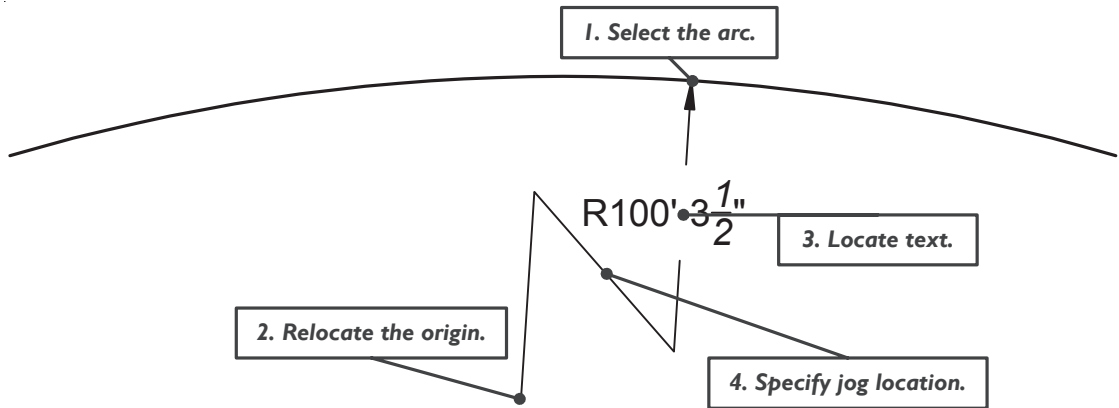
Dimension text = 10.0

Specify dimension line location or [Mtext/Text/Angle]: *(Pick a point to position the dimension.)*

The **Mtext**, **Text**, and **Angle** options are the same as those described earlier in this chapter under the DIMHORIZONTAL command.

DIMJOGGED

The DIMJOGGED command draws radial dimensions with a jogged leader line (also called “foreshortened” radius dimensions).



This command has more options, because AutoCAD also needs to know (1) where to relocate the dimension origin, and (2) where to place the jog.

The relocated origin replaces the center point, even though it should always point to the center. The jog location determines the middle point of the jog; the angle of the jog (called the “transverse angle”) is determined by the dimension style.

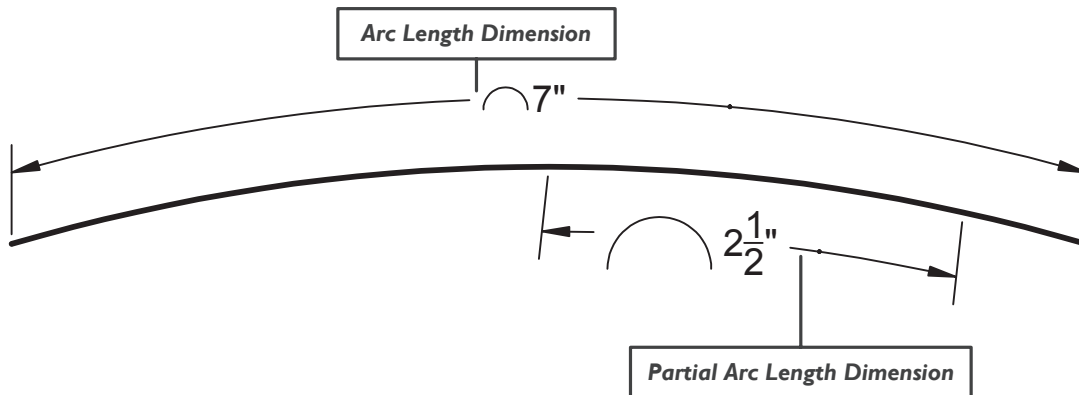
TUTORIAL: PLACING JOGGED RADIAL DIMENSIONS

- To dimension a radius with a jogged leader, start the **DIMJOGGED** command:
 - At the 'Command:' prompt, enter the **dimjogged** command.
Command: **dimjogged** (Press ENTER.)
 - Alternatively, enter the aliases **jog** or **djo** at the 'Command:' prompt.
- In all cases, AutoCAD prompts you to select the arc or circle you want dimensioned:
Select arc or circle: (Pick one arc, circle, or polyarc.)
- Pick a point to relocate the origin point closer to the arc:
Specify center location override: (Pick a point.)
Dimension text = 101'-3 1/2"
- Indicate the center of the dimension text:
Specify dimension line location or [Mtext/Text/Angle]: (Pick a point or enter an option.)
- Locate the jog's center:
Specify jog location: (Pick a point.)



The **DIMARC** command measures the lengths of arcs.

This command places dimensions along the circumference of arcs. The dimension can be along the full length of the arc, or just a portion.



TUTORIAL: PLACING ARC LENGTH DIMENSIONS

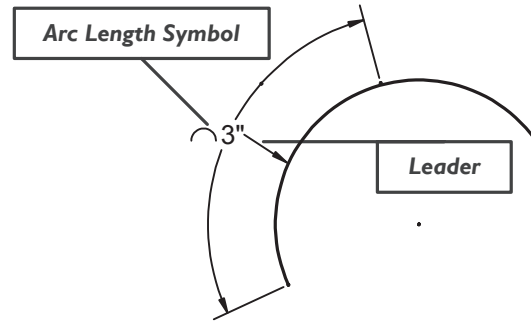
- To dimension the length of an arc, start the **DIMARC** command:
 - At the 'Command:' prompt, enter the **dimarc** command.
Command: **dimarc** (Press ENTER.)
 - Alternatively, enter the **dar** alias at the 'Command:' prompt.
- In all cases, AutoCAD prompts you to select an arc or polyline arc:
Select arc or polyline arc segment: (Pick one arc or polyarc.)
- Indicate the position of the dimension text:
Specify arc length dimension location, or [Mtext/Text/Angle/Partial/Leader]: (Pick a point or enter an option.)

ARC LENGTH DIMENSIONS: ADDITIONAL METHODS

The DIMARC command has three options you are already familiar with — **Mtext**, **Text**, and **Angle**. Options unique to arc lengths are:

- **Partial** measures parts of arc lengths.
- **Leader** toggles the addition of leaders.

The figure below shows the effect of the partial and leader options.



Partial

The **Partial** option dimensions just part of the length of the arc. AutoCAD prompts you to pick two points along the arc between which the dimension is placed:

Specify first point for arc length dimension: *(Pick a point on the arc.)*

Specify second point for arc length dimension: *(Pick another point.)*

If **ENDpoint** osnap is on, this command will snap to the endpoints of the arc, negating the attempt at partial dimensioning.

The results differ, depending on where you place the partial dimension. If the dimension is between extension lines, the extensions lines are parallel to each other; when the dimension is outside, the extension lines are radial to the arc's center point.

Leader

The **Leader** option toggles (turns on and off) the addition of a leader to arcs greater than 90 degrees. (This option does not appear when the arc is less is 90 degrees.) The leader points from the dimension text to the arc.

After entering the **Leader** option, the prompt changes to **No leader**:

Specify arc length dimension location, or [Mtext/Text/Angle/Partial/Leader]: **l**

Specify arc length dimension location, or [Mtext/Text/Angle/Partial/No leader]: *(Enter option.)*

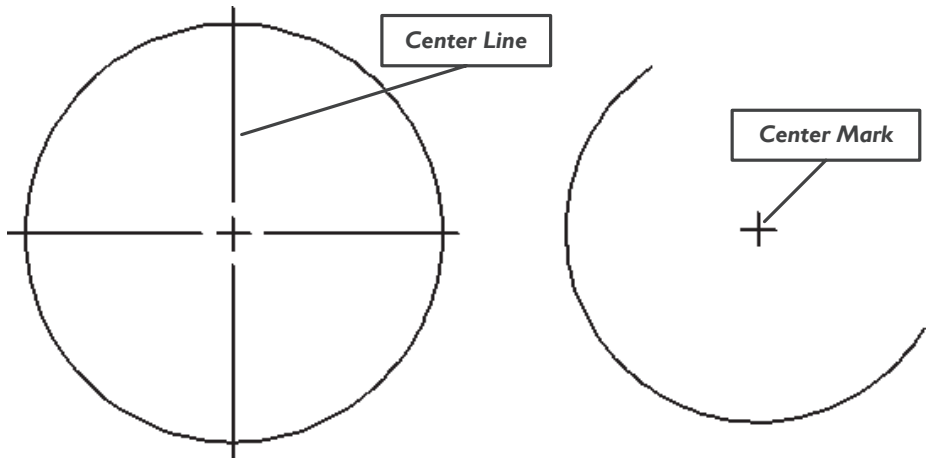


DIMCENTER

The DIMCENTER command places center marks and lines on arcs and circles.

TUTORIAL: PLACING CENTER MARKS

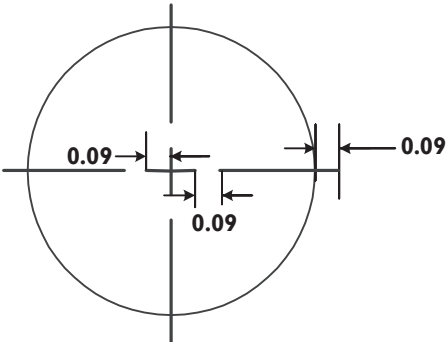
- To mark the center of arcs and circles, start the **DIMCENTER** command:
 - In the ribbon's Annotation tab, choose the **Center Mark** button in the Dimensions panel.
 - At the 'Command:' prompt, enter the **dimcenter** command.
Command: **dimcenter** *(Press ENTER.)*
 - Alternatively, enter the **dce** alias at the 'Command:' prompt.



- 2. In all cases, AutoCAD prompts you to select the arc or circle you want dimensioned:
Select arc or circle: *(Pick one arc, circle, or polyarc.)*
Notice that AutoCAD places the center mark.

DimCen

The DIMCEN system variable defines the size and look of the center mark. This variable affects center marks created by DIMRADIUS and DIMDIAMETER, although their center marks are drawn only when the dimension is placed outside the circle or arc.



The default value is 0.09, which means that DIMCENTER draws the center mark lines 0.09 units long.

DimCen	Meaning
0	Draws neither center lines nor center marks.
<0	Draws center lines and center marks.
>0	Draws center marks only.

When the value is negative, such as -0.09, AutoCAD leaves a gap of 0.09 units between the mark and the line, as well as extends the line 0.09 units beyond the circumference of the circle or arc.

To draw center lines as well, change the value of DIMCEN to -0.09 (the negative value), as follows:

```
Command: dimcen
Enter new value for DIMCEN <0.0900>: -0.09
```

Normally, this value is set in the dimension style using the DIMSTYLE command.

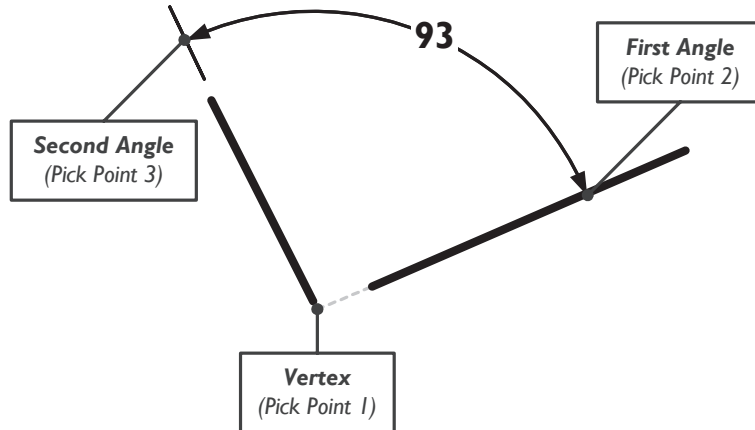


DIMANGULAR

The DIMANGULAR command dimensions angles, arcs, circles, and lines.

Unlike the other dimensioning commands in this chapter, this dimensions angles only — not lengths or diameters. The command requires a *vertex* and some means of determining the angle. The result depends on the object selected:

Two Lines — the vertex is the real or apparent intersection of the two lines; the angle is measured between the two lines.

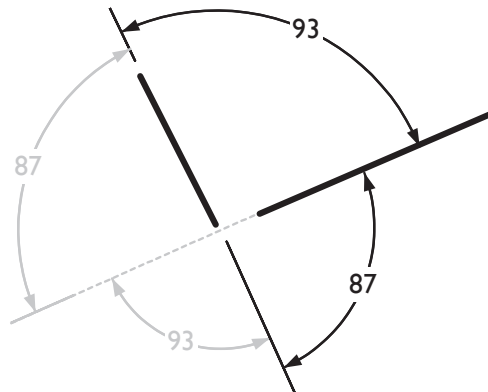


Arc — the vertex is the arc's center point; the angle is measured between two endpoints.

Circle — the vertex is the circle's center point; the angle is measured between the “Select circle” pick and the next pick point.

Three Points — the vertex is at the first pick point; the angle is measured between the next two pick points.

For intersecting lines, DIMANGULAR can draw four possible angles, each a supplementary pair adding up to 180 degrees. In the figure below, the supplementary pairs are shown in the same color.



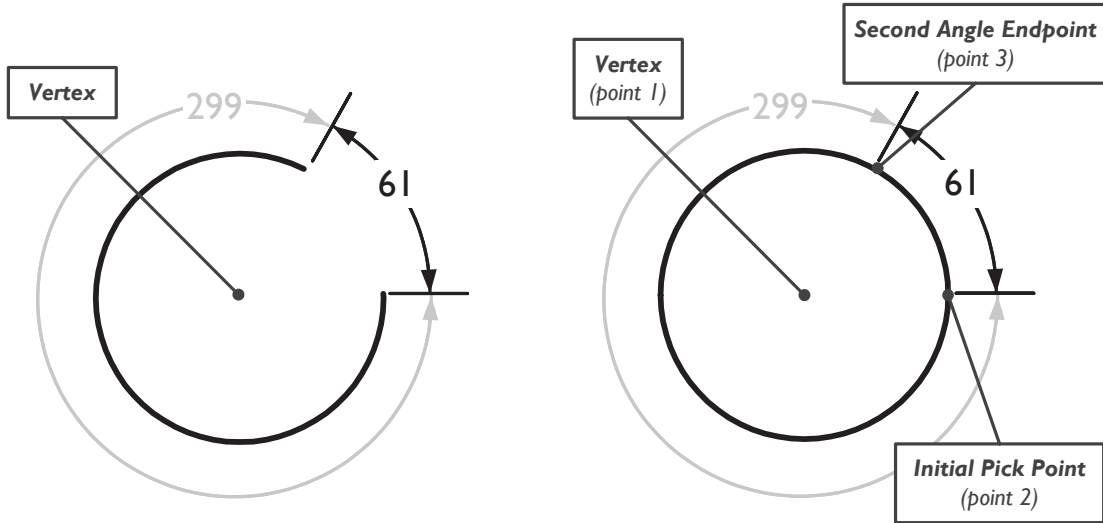
In the case of arcs and circles, two angles are possible: the *minor arc* (shown in black, below) and the *major arc* (shown in gray). Together they add up to 360 degrees.

TUTORIAL: PLACING ANGULAR DIMENSIONS

- I. To dimension an angle, start the **DIMANGULAR** command:
 - In the ribbon's Annotation tab, choose the **Angular** button in the Dimensions panel.
 - At the 'Command:' prompt, enter the **dimangular** command.

Command: **dimangular** (Press ENTER.)

- Alternatively, enter the aliases **dan** or **dimang** at the 'Command:' prompt.



- In all cases, AutoCAD prompts you to select the object you want dimensioned:
Select arc, circle, line, or <specify vertex>: (Press ENTER for the vertex option)
- Pick three points to specify the angle:
Specify angle vertex: (Pick a vertex at point 1.)
Specify first angle endpoint: (Pick point 2.)
Specify second angle endpoint: (Pick point 3.)
Non-associative dimension created.
- Pick a point to locate the dimension:
Specify dimension arc line location or [Mtext/Text/Angle/Quadrant]: (Pick a point to position the dimension.)
Dimension text = 90

The **Mtext**, **Text**, and **Angle** options are the same as those described earlier in this chapter under the **DIMHORIZONTAL** command. The unique option is **Quadrant**.

Quadrant

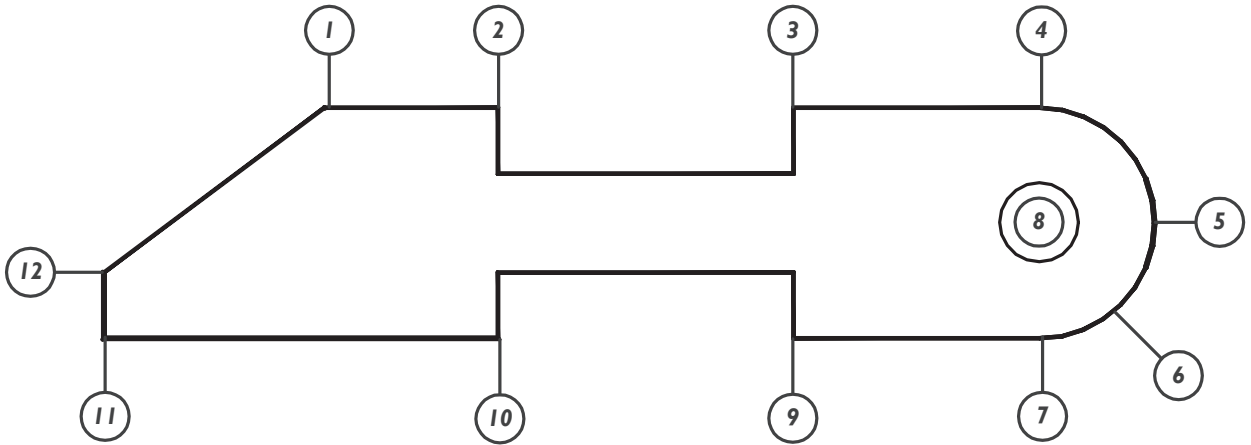
The **Quadrant** option forces the angular dimension to reside in one of the four quadrants:

Specify quadrant: (Drag the dimension around the arc.)

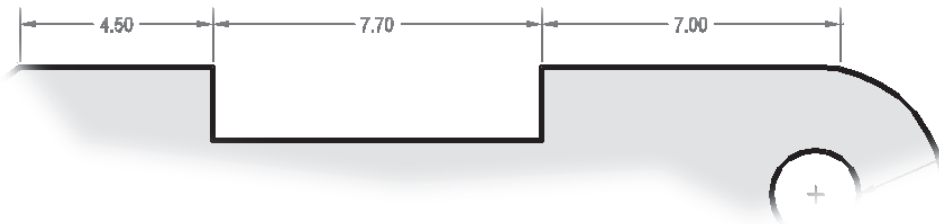
Pick a quadrant for the dimension. When you drag the dimension beyond the end of the arc, AutoCAD adds an extension, just as with the **DIMRADIUS** command.

TUTORIAL: DIMENSIONING

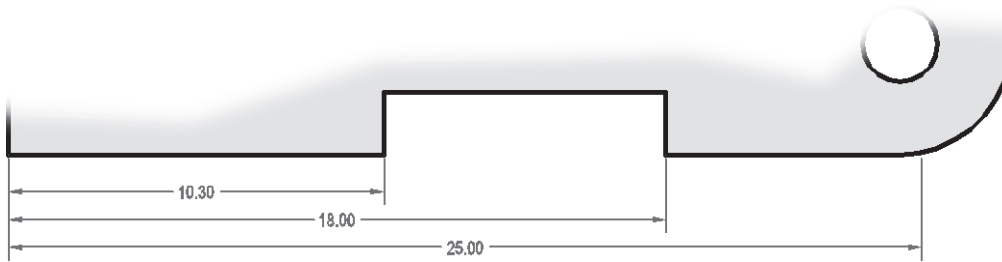
In the following exercise, you dimension the fuse link below with some of the commands learned in this chapter. As you work through the tutorial, refer to the numbered points in the figure below.



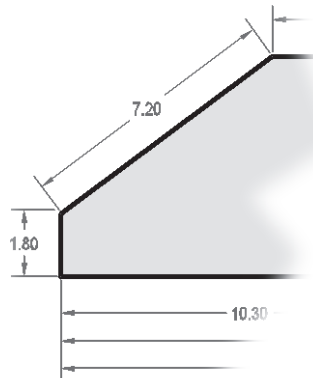
1. From the companion CD, open the *dimen.dwg* file, a drawing of a fuse link. This drawing has layers, dimension styles, and object snaps preset for you. Check the Layers toolbar to ensure layer “Dimensions” is current.
2. Start by placing a linear dimension along the top of the fuse link. From the Dimension menu, select **Linear**. Press **ENTER** to use the **Select Object** option, and then select the line between points 1 and 2. Locate the dimension line roughly 0.5 units above.
3. Continue dimensioning with the **DIMCONTINUE** command. Use **INTersection** object snap at points 3 and 4. Complete the dimension at point 5; do not dimension the arc at this time. Remember to press **ESC** to exit the command. Do the new dimension lines continue at the same height as the first dimensions?



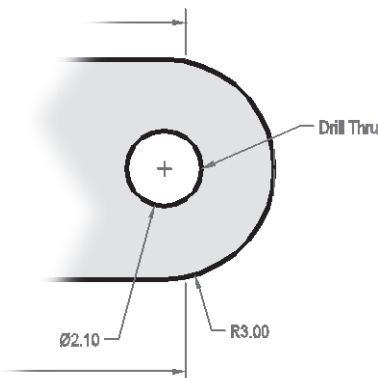
4. Now start a linear dimension along the bottom of the fuse link with the **DIMLINEAR** command. Place extension lines at points 11 and 10. Does **INTersection** object snap help you?
5. Switch to baseline dimensioning. From the Dimension menu, choose **Baseline**. Pick points 9 and 7 as the second extension line origins. Complete the command by picking point 7; do not dimension the arc yet.



6. Place a vertical dimension by entering the **DIMLIN** alias.
Choose point I1 as the first extension line origin, and point I2 as the second.
If you make a mistake, use the **UNDO** command to eliminate the last dimension.
7. Dimension the angle at the left side with the **DIMALIGNED** command.
Use the **Select Object** option to select the angled line between points I2 and I1.

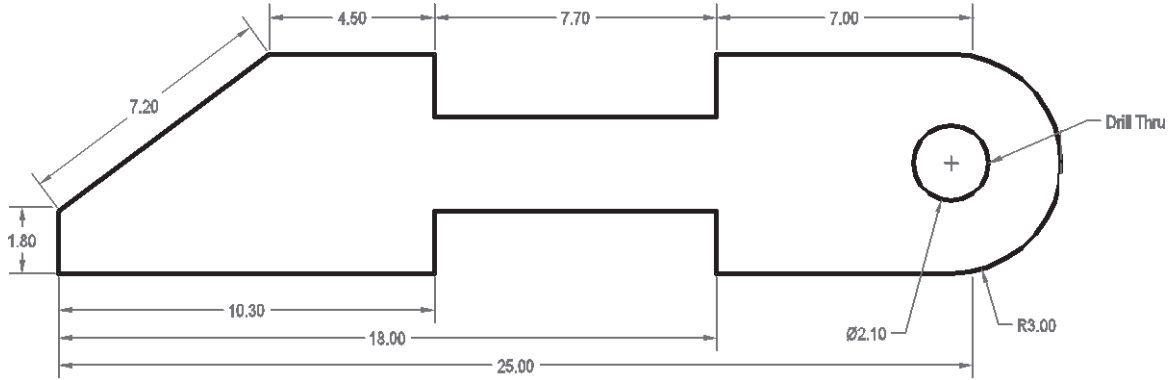


8. Select **Diameter** from the **Dimension** menu.
Dimension the circle at point 8.
Make sure the leader line extends outside the fuse link.



9. Dimension the arc with **DIMRADIUS**.
Pick point 6, and then place the dimension leader outside the fuse link.
10. Construct a leader by entering the **QLeader** command. (See Chapter 10, "Placing and Editing Text," for more about leaders.)
Start the leader at the circle (point 8), and end it outside the object.
For the leader text, enter "Drill Thru."

Remember to press **ENTER** to exit the **QLEADER** command.



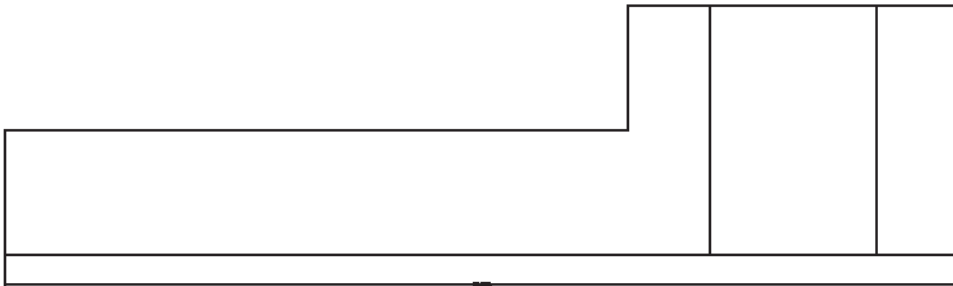
11. Save the completed drawing as *dimen.dwg*.

TUTORIAL: ANNOTATIVE DIMENSIONING

Your boss has emailed you *Anno-Dim.dwg*, requesting that you dimension it. In his note, he remarks that the drawing will be needed in detail sheets with two different scale factors, one at 1:8 scale and the other at 1:16.

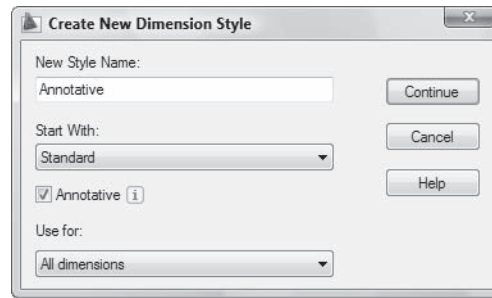
In this tutorial, you use annotative scaling to dimension the drawing.

1. From Companion CD, open the *Anno-Dim.dwg* file.
2. On the status bar, change the **Annotation Scale** to 1:8, the first value your boss specified.
3. You start the **DIMLINEAR** command:
 Command: **dimlinear**
 Specify first extension line origin or <select object>: (Press **ENTER**.)
 Select object to dimension: (Select a line.)
 Specify dimension line location or [Mtext/Text/Angle/Horizontal/Vertical/Rotated]: (Pick a point.)

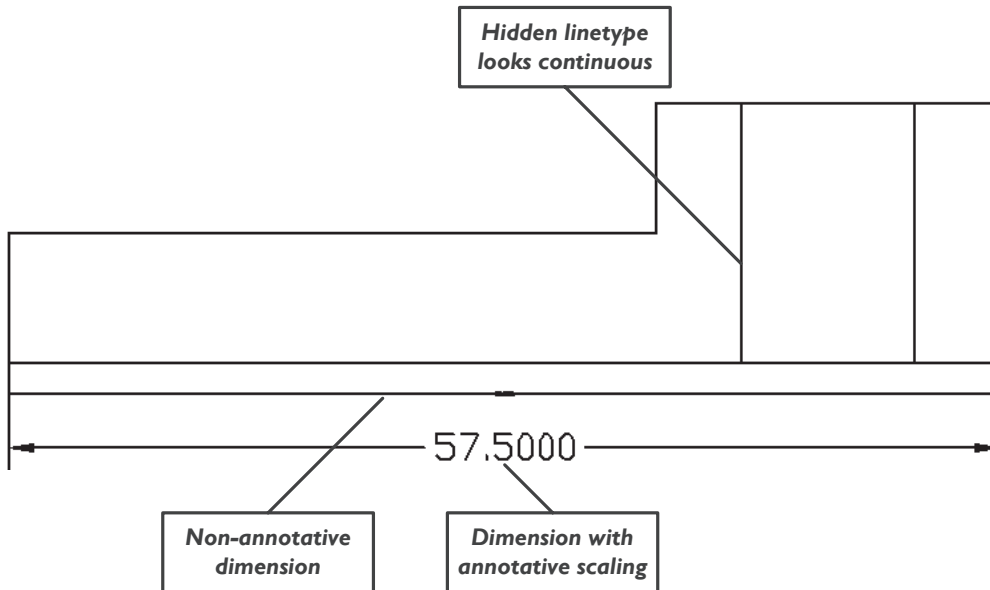


... but find that the dimension elements are too small. Of course, you should be using annotative scaling.

4. You note that the drawing file lacks an annotative dimension style, and so you create one.
 - a. In the ribbon's Dimensions panel, click Dimension Style button
 - b. Click **New**, and then name it "Annotative."
 - c. Ensure that the **Annotative** option is turned on, and then click **Continue**.
 - d. You don't need to change anything else, so click **OK**, **Set Current**, and **Close**.
 The new annotative dimension style is now the current one.



5. Reuse the **DIMLINEAR** command. That's better!



Notice that annotative scaling takes over the duty of the **DIMSCALE** system variable.

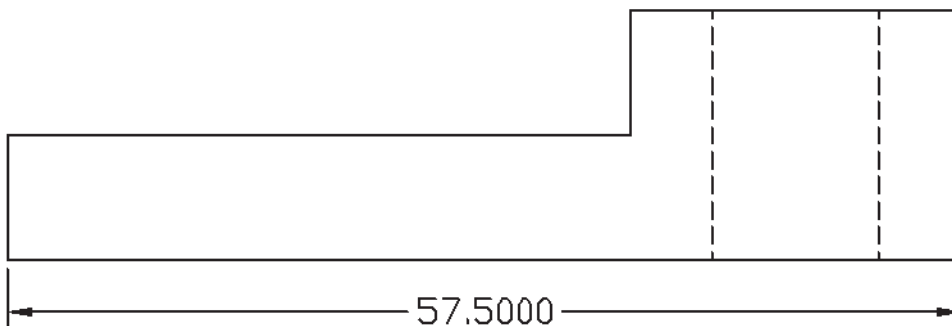
6. Erase the first linear dimension, the one that's scaled too small.
7. The two vertical lines should be showing hidden lines, but they look continuous. The Properties palette confirms that they have the Hidden linetype, but are scaled incorrectly. If necessary, change the value of **MSLTSCALE** to 1, and then use the **REGEN** command to update the drawing.

Command: **msltscale**

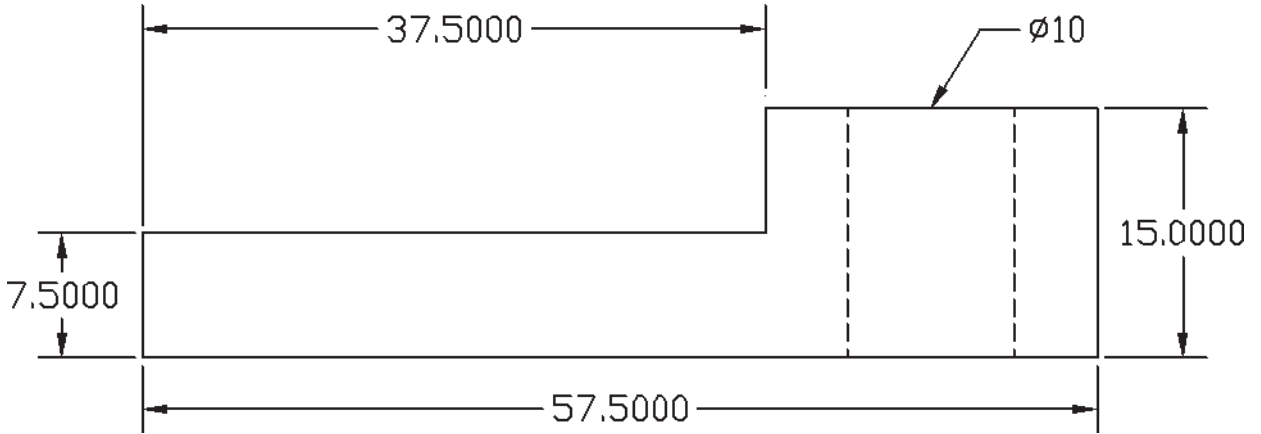
Enter new value for MSLTSCALE <0>: **1**

Command: **regen**

Notice how annotative scaling takes care of scaling linetypes correctly.

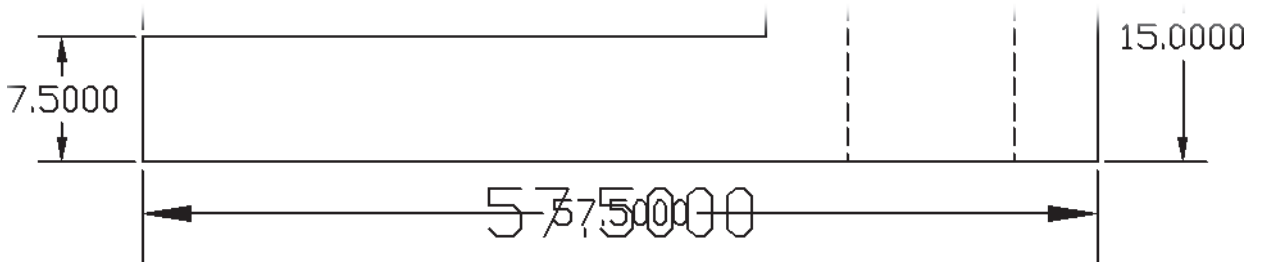


8. Add the other dimensions required by the drawing.



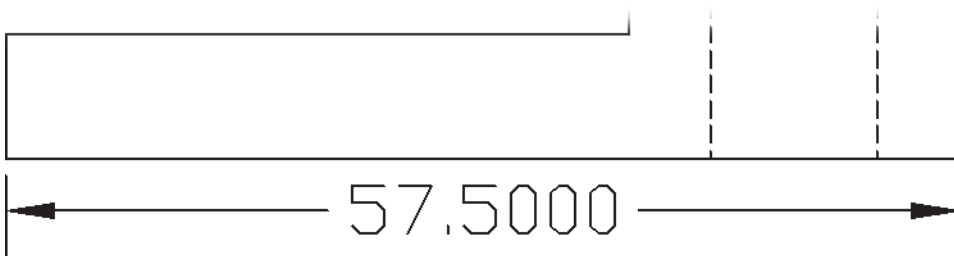
9. The other scale your boss wanted was 1:16. Change **Annotation Scale** to 1:16.

10. Re-run the **DIMLINEAR** command.



Notice that the dimension text and arrowheads are twice as large — annotative scaling takes care of that.

11. But, oh no! You can see both scaled versions of the linear dimension. The quick way to deal with that is to turn off **ANNOALLVISIBLE**: click the **Annotation Visibility** button on the toolbar.



That's better.

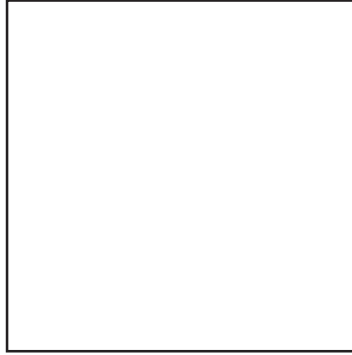
12. Finish dimensioning the drawing, and then save it.

An alternative approach is to apply one set of dimensions at one annotative scale. Once applied, select all, and then use the Properties palette to add additional annotative scale factors.

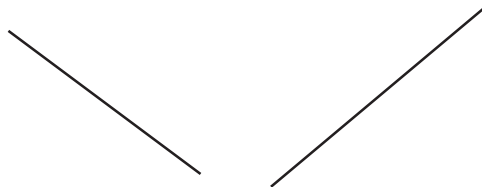
The technical editor suggests that the better practice is to use the Properties palette to add the second scale to existing dimensions.

EXERCISES

1. Start a new drawing.
With the **POLYGON** command, draw a square approximately one-third the size of the screen.
Use the **DIMLINEAR** command along with object snap **INTersection** to capture the lower left and lower right corners of the square.
Place the dimension line below the bottom line of the square.

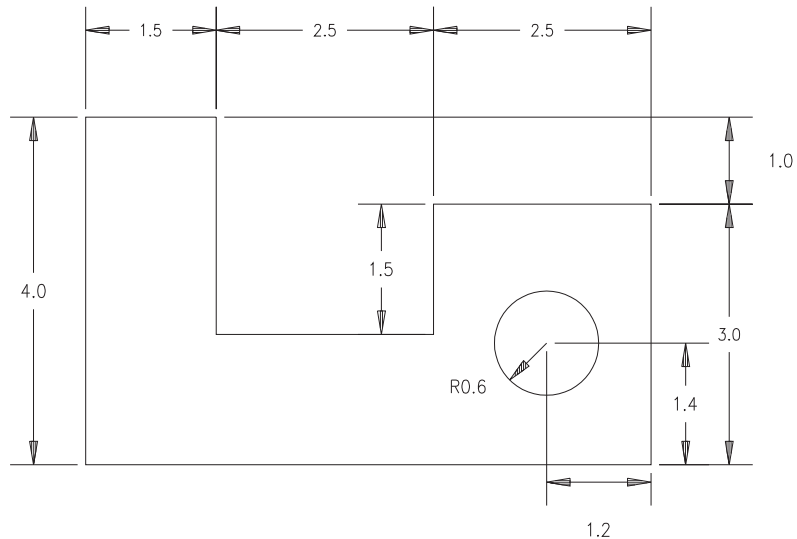


2. Continuing from the previous exercise, repeat the command for the upper line of the square, with a change. Instead of picking the corners, dimension the line by selecting it.
Is the dimension text the same for the lower and upper lines?
3. Continuing, dimension one of the vertical sides of the box.
Is the vertical dimension the same as the horizontal dimension?
4. Next, dimension the remaining vertical line.
At the 'Specify dimension line location or [Mtext/Text/Angle/Horizontal/Vertical/Rotated]' prompt, enter **m**.
Does the mtext editor dialog box appear?
Type your name.
Choose **OK** when finished.
What appears along the dimension line?
Save the drawing as *square1-4.dwg*.
5. Start a new drawing.
Draw a scalene triangle.
Use the **DIMALIGNED** command to dimension the three angled sides.
Does AutoCAD draw the dimension lines parallel to the sides?
6. Draw two lines that are not parallel, similar to those shown below.
Place an angular dimension on the two lines.

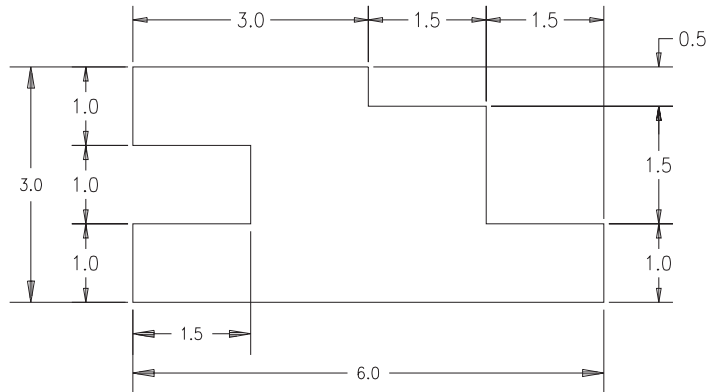


- Repeat the command to draw the supplemental angle.
7. Draw an arc and a circle.
Dimension the minor and major arcs with the **DIMANGULAR** command.
Save the drawing as *dimen5-7.dwg*.

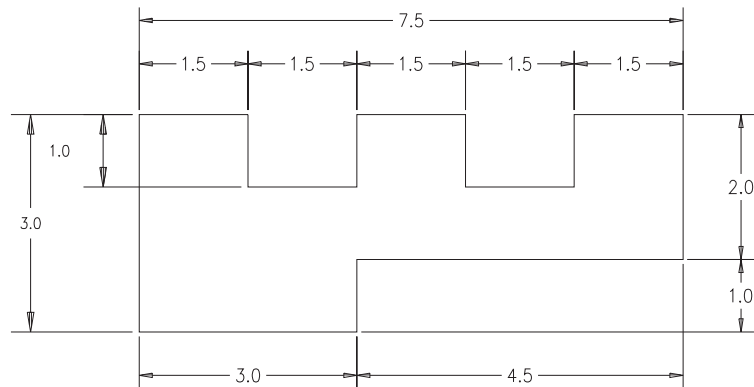
8. Start a new drawing.
 Draw a diagonal line, similar to the ones in exercise #6.
 Use the following commands to dimension the line three times: horizontally, vertically, and rotated.
 - a. **DIMHORIZONTAL**
 - b. **DIMVERTICAL**
 - c. **DIMROTATED**
9. Undo the three dimensions you drew in the previous exercise.
 Use the **DIMLINEAR** command to draw the same three orientations of dimension.
 Which commands did you find easier to use?
 Save the drawing as *linear8-9.dwg*.
10. Start a new drawing.
 Draw an arc and a circle on the screen, and then dimension each with the **DIMDIAMETER** command.
 Practice placing diameter dimensions inside and outside the circle, and at different locations around the arc.
11. Repeat exercise #9 with the **DIMRAD** command.
 Save the drawing as *diarad10-11.dwg*.
12. From the companion CD, open the *qdim.dwg* drawing of a flywheel.
 Dimension the two views with the **QDIM** command.
 When complete, save the drawing as *flywheel12.dwg*.
13. Draw the following baseplates, and then place the dimensions at the locations shown.
 - a. Save the drawing as *baseplate13-a.dwg*.



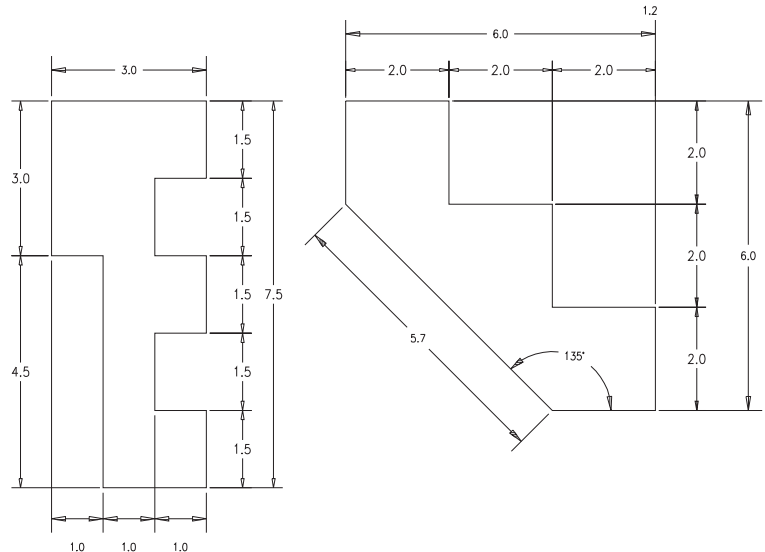
b. Save the drawing as *baseplate13-b.dwg*.



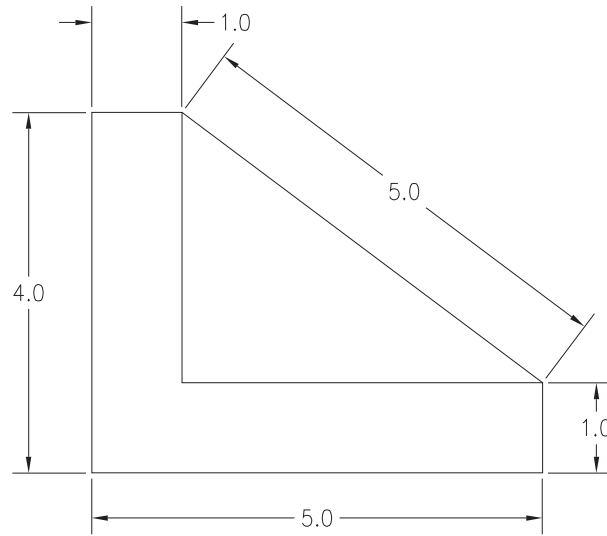
c. Save the drawing as *baseplate13-c.dwg*.



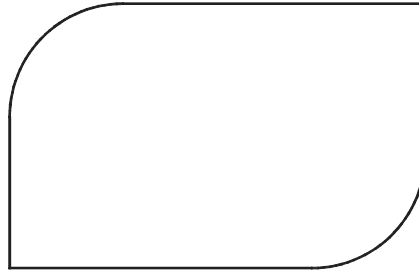
14. Draw the two views of the clamp, and then use dimension commands to place the vertical dimensions at the locations shown. Save the drawing as *clamp14.dwg*.



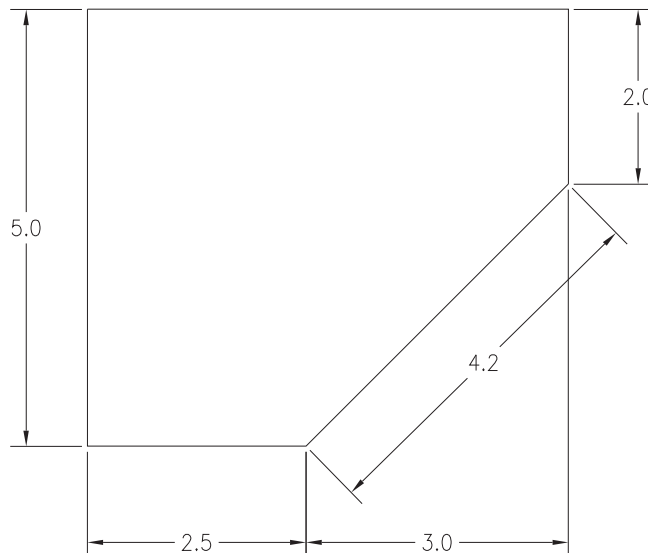
15. Construct the drawing of the corner shelf, and then place all the dimensions as shown. Save the drawing as *shelf15.dwg*.



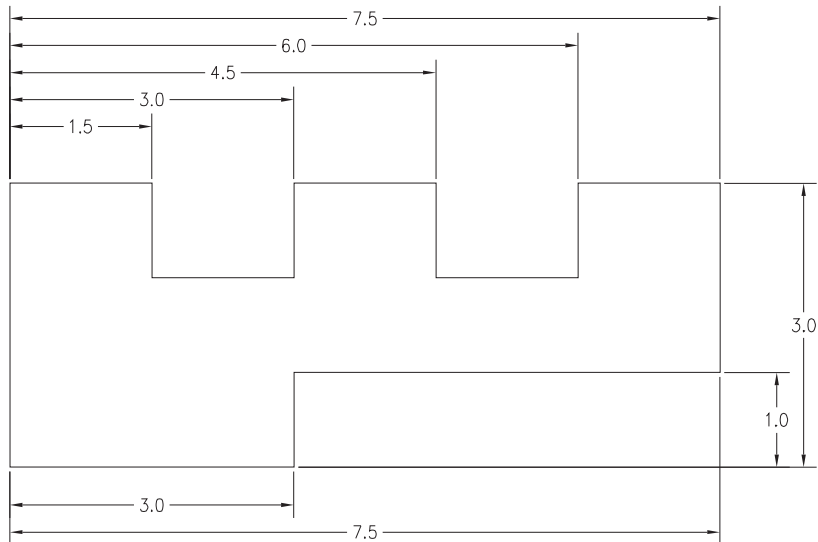
16. Draw the following 5.5" x 3.5" object, and then use radius dimensioning to specify the two 1.5" fillets. Save the drawing as *radial16.dwg*.



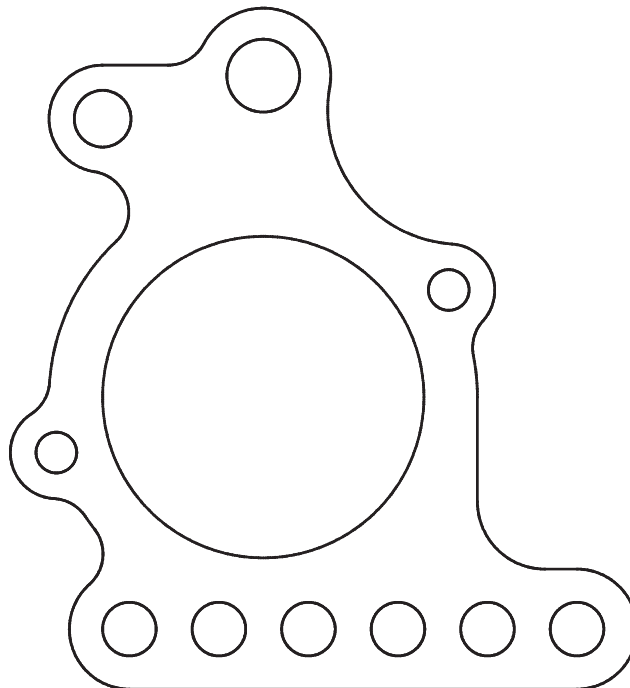
17. Construct the drawing of the corner shelf, and then place all the dimensions as shown. Save the drawing as *shelf17.dwg*.



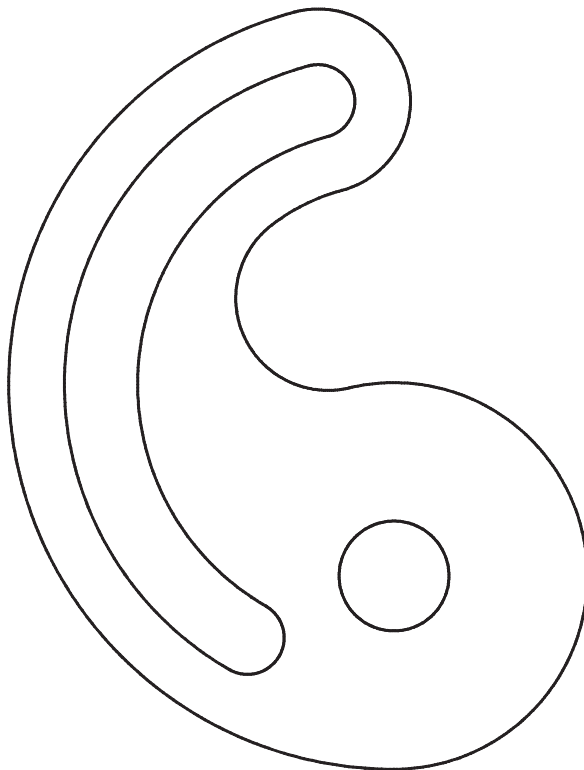
18. Construct the drawing of the jig, and then place the baseline dimensions.
Save the drawing as *jig18.dwg*.



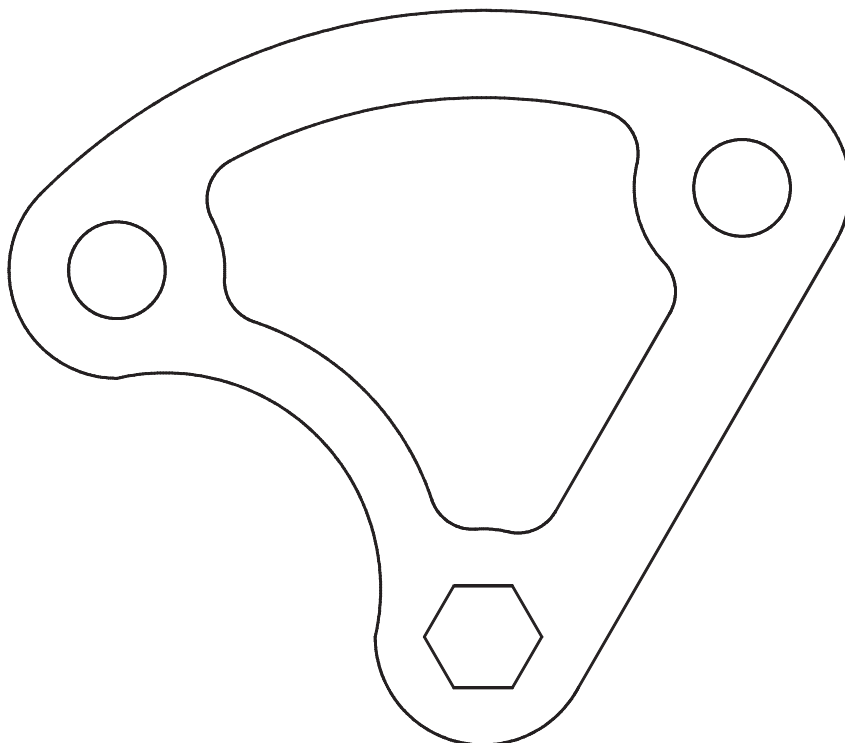
19. From the companion CD, open the *15-47.dwg* of a gasket.
Dimension the drawing with the **DIMRAD** and **DIMDIA** commands.
When complete, save the drawing by the same name.



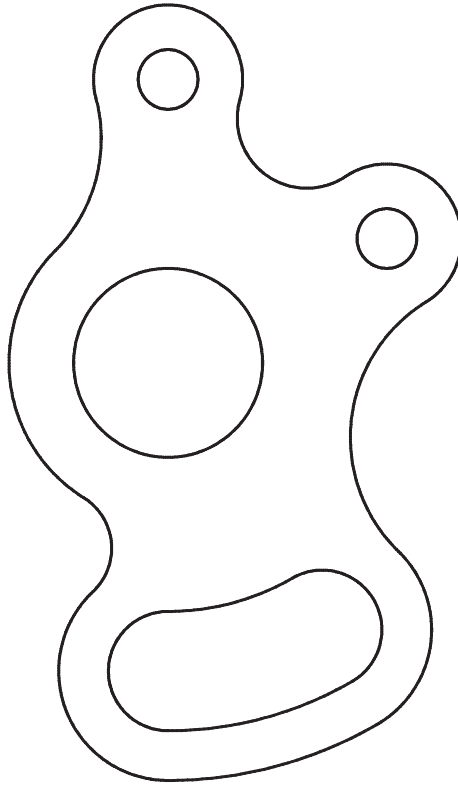
20. From the companion CD, open the following drawings, and then dimension the gaskets.
- Save as 5-43.dwg.



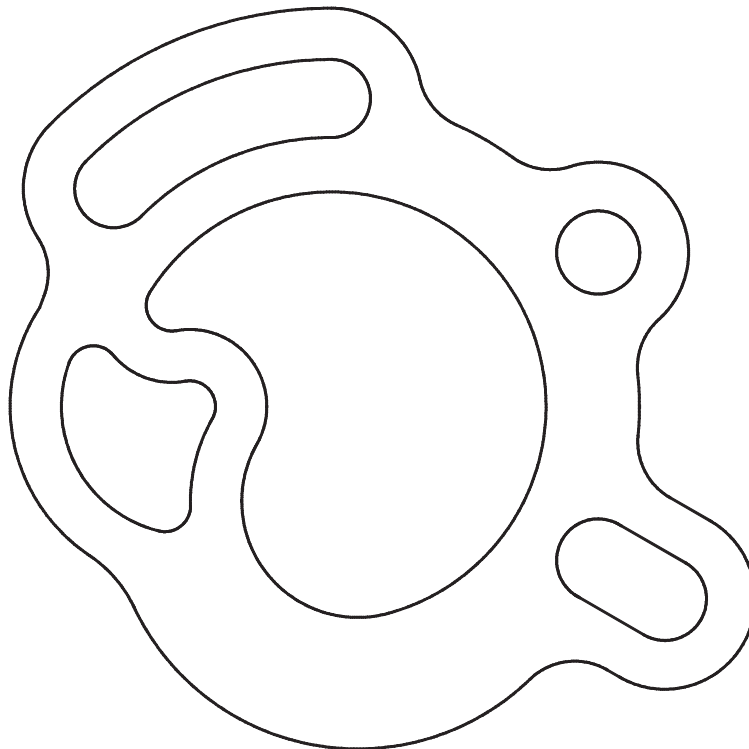
- Save as 15-44.dwg.



c. Save as 15-45.dwg.

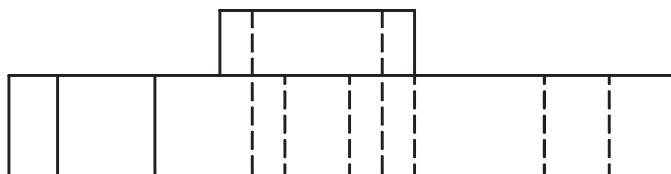
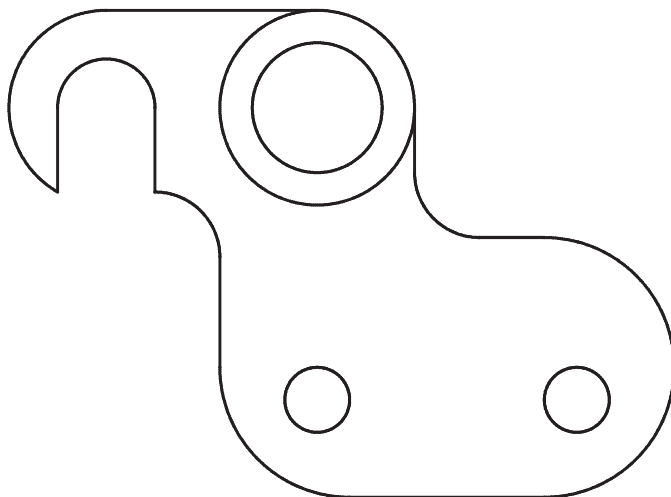


d. Save as 15-46.dwg.

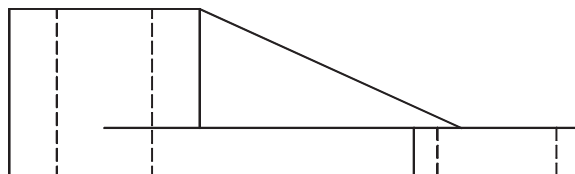
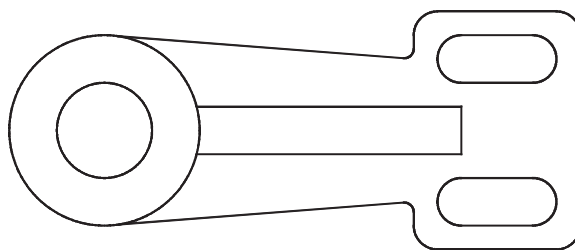


21. From the Companion CD, open the following drawings. Dimension each drawing, and then save your work.

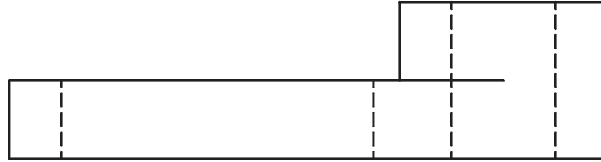
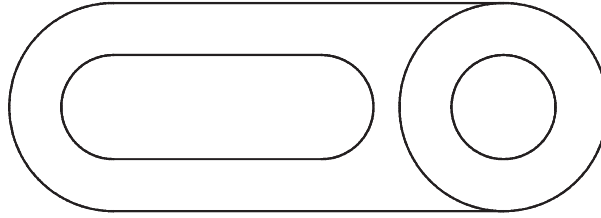
a. 17_19.dwg: Attachment for a three-point hitch.



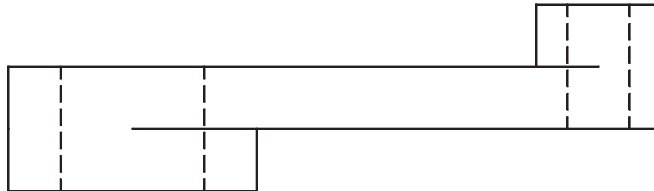
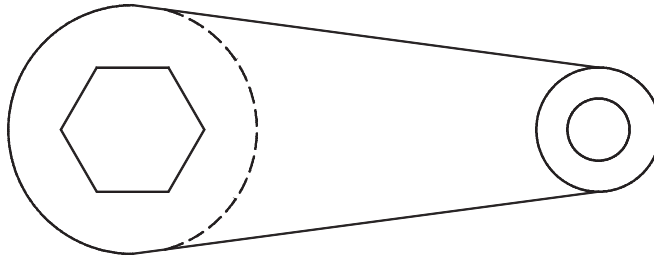
b. 17_20.dwg: Flag pole stabilizer.



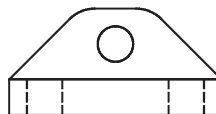
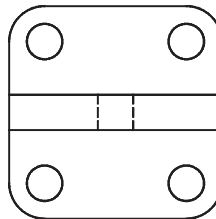
c. 17_21.dwg: Reciprocating linkage.



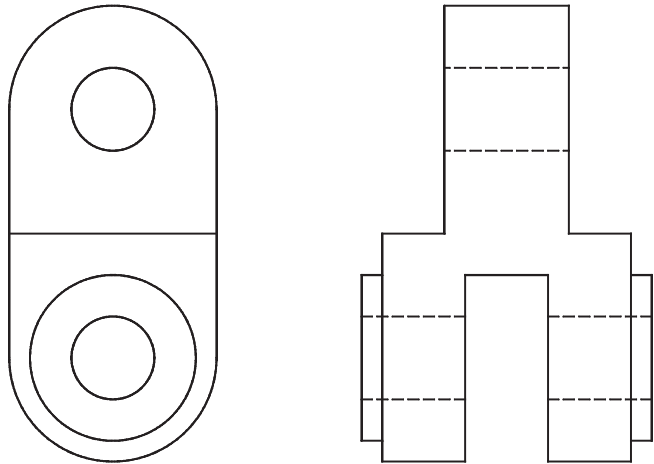
d. 17_22.dwg: Socket linkage.



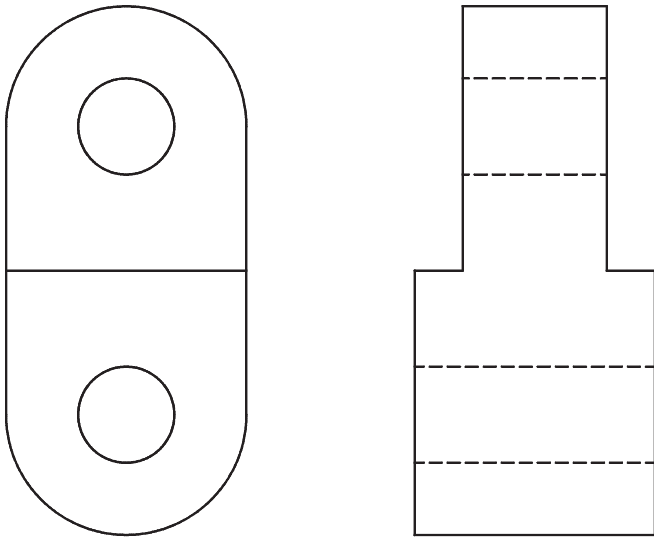
e. 17_23.dwg: Attachment plate.



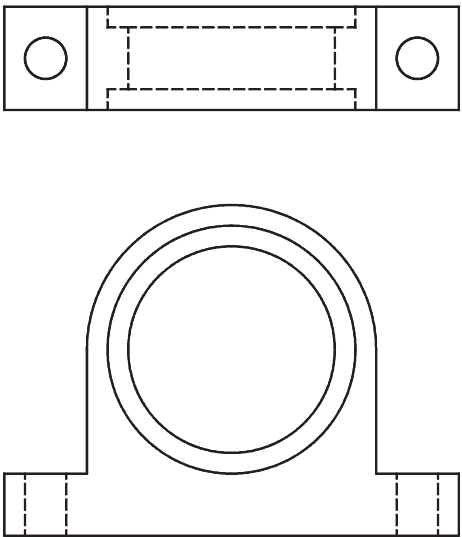
f. 17_24.dwg: Hitch yoke.



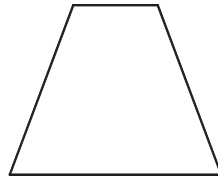
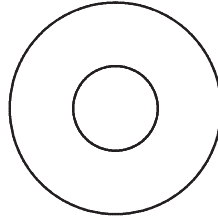
g. 17_25.dwg: Hitch linkage.



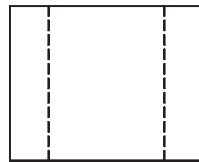
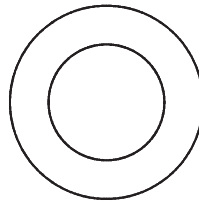
h. 17_26.dwg: Axle support.



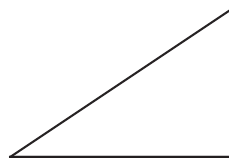
- i. 17_27.dwg: Cone of silence. (Technical editor comments: "Yes! Maxwell Smart lives!"
Copy editor comments: "Sorry about that, Chief.")



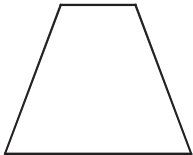
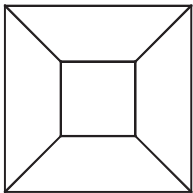
- j. 17_28.dwg: Bushing.



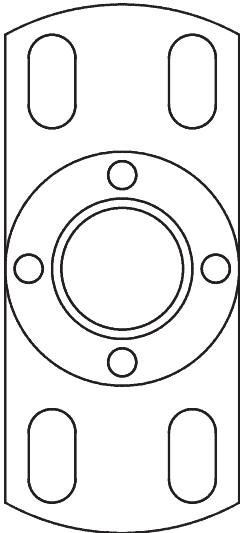
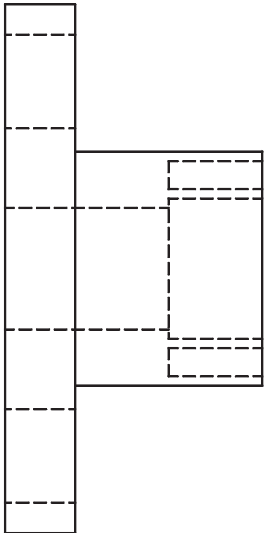
- k. 17_29.dwg: Wedge.



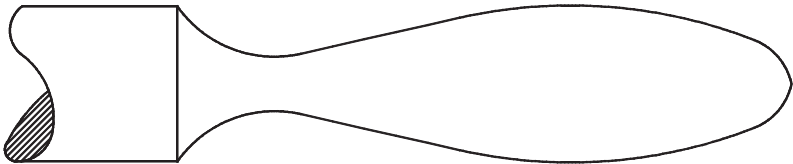
l. 17_30.dwg: Concrete footing.



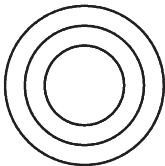
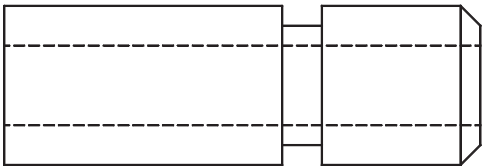
m. 17_31.dwg: Heavy-duty axle support.



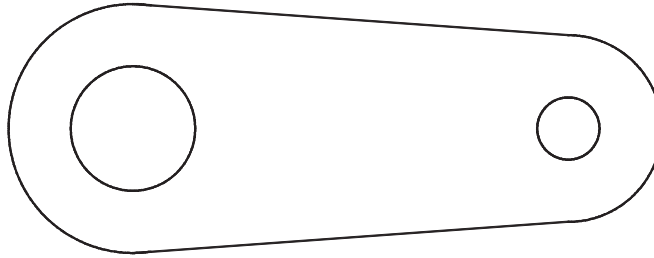
n. 14_32.dwg: Paintbrush handle.



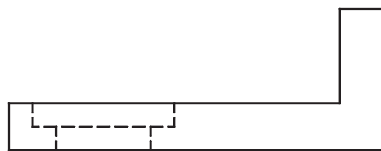
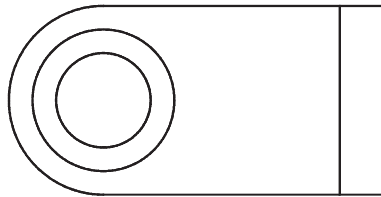
o. 17_33.dwg: Spacer.



p. 17_34.dwg: Baseplate.



q. 17_35.dwg: Angled baseplate. Use the **QDIM** command.



CHAPTER REVIEW

1. A dimension measures _____.
2. Should extension lines touch the objects they reference?
3. Where should dimensions be placed, whenever possible?
4. Leaders indicate _____.
5. How many possible angles can **DIMANGULAR** draw on:
 - Circles?
 - Pair of lines?
6. Should you attempt to dimension every detail?
7. Under what conditions are dimension lines not drawn?
8. What is another name for a “witness line”?
9. How are the endpoints of extension lines determined?
10. Must a dimension always have two extension lines?
11. Sketch and label the parts of a measurement dimension. Include the following parts:
 - Dimension line
 - Arrowheads
 - Extension lines
 - Text
12. Sketch and label the parts of a radial dimension. Include the following parts:
 - Leader line
 - Arrowhead
 - Text
 - Radius Symbol
13. Sketch and label the parts of an extension line:
 - Extension line
 - Offset
 - Extension beyond dimension line
14. List the types of linear dimensions that the **DIMLINEAR** command draws.
 - a.
 - b.
 - c.
15. Explain the function of the **DIMDIAMETER** command?
16. What is the purpose of the **QDIM** command?

17. Sketch an example of a baseline dimension.

18. Must arrowheads always be arrows?

19. What is the aspect ratio of the standard arrowhead?

20. List three places where dimension text can be placed.

21. Must dimension text always be horizontal?

22. What angle does the **DIMANGULAR** command measure for the following objects:

Circle

Two Lines

Arc

23. Explain why *direct dimensioning* is beneficial.

Can any object be dimensioned directly?

24. Which dimensioning commands do you use most often with arcs and circles?

25. What is the purpose of the **DIMANGULAR** command?

26. Describe the meaning of “fully associative dimensioning.”

27. Are all dimensions associative?

Give an example.

28. How is the look of dimensions controlled?

29. Where in AutoCAD might you find dimension standards?

30. Name three object snaps useful for dimensioning:

a.

b.

c.

31. Explain the purpose of the center mark.

Describe the difference between a center mark and a center line.

How do you instruct AutoCAD to draw one or the other?

32. Explain the meaning of the following options available in many dimensioning commands:

Angle

Mtext

Text

33. What is the **<>** symbol meant for?

34. A dimension reads 2.5. Write out the text string that changes the dimension to read:

a. “Adjust to 2.5m”

b. “Turn by 2.5° increments.”

c. “Temperature range is 2.5°±0.5°”

35. Which part of an arc does the **DIMALIGNED** command measure?
Of a circle?
Of a line?
36. How do these commands differ?
DIMBASELINE
DIMCONTINUE
37. Can you apply the **DIMCONTINUE** command to a diameter dimension?
38. Write out the command for each alias:
dra
dco
dli
dimali
39. Can you dimension ellipses and splines with the **DIMRADIUS** command?
40. The **DIMJOGGED** command is best suited for which kinds of arcs and circles?
41. When would you use the **DIMARC** command?



Editing Dimensions

You often need to change existing dimensions. This chapter describes how to edit the position and properties of dimensions. After completing this chapter, you will have an understanding of these commands:

AIDIMFLIPARROW flips the direction of arrowheads.

DIMJOGLINE adds jogs to dimension lines.

DIMBREAK adds breaks to dimension and extension lines.

DIMSPACE spaces groups of dimensions evenly.

DIMEDIT slants extension lines.

DIMTEDIT relocates dimension text.

DDEDIT changes dimension text with the mtext editor.

DIMINSPECT adds and removes inspection text.

AIDIM and **AI_DIM** prefix a group of undocumented dimension editing commands.

DIMSTYLE creates and modifies dimension styles.

DIMOVERRIDE applies changed dimvars to selected dimensions.

TEXTTOFRONT changes the display order of dimensions.

Grips edit dimensions and leaders directly.

NEW TO AUTOCAD 2010 IN THIS CHAPTER

- Dimension text can be placed under the dimension line.
- The reading order of dimension text can be reversed.
- Subunits can now be specified.



AIDIMFLIPARROW

The AIDIMFLIPARROW command flips the direction of arrowheads.

This command flips the direction of selected arrowheads by 180 degrees, which is useful for touching up dimensions when arrowheads would look better placed outside the extension lines, or vice versa. It is not documented by Autodesk, but appears in the shortcut menu when a dimension is selected.

TUTORIAL: FLIPPING ARROWHEADS

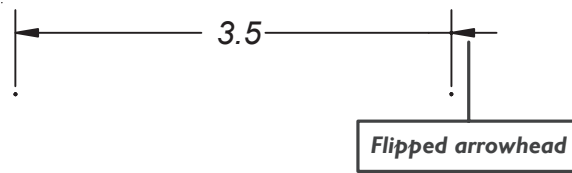
1. Draw a linear dimension with the **DIMLINEAR** command.
2. To change the direction of an arrowhead, start the **AIDIMFLIPARROW** command:
 - Select the dimension, right-click, and then choose **Flip Arrow** from the shortcut menu.
 - At the 'Command:' prompt, enter the **aidimfliparrow** command.

Command: **aidimfliparrow** (Press ENTER.)

3. AutoCAD prompts you to select objects, but you really can choose only one arrowhead at a time.

Select objects: (Select one or more arrowheads.)

AutoCAD flips the arrowhead, and draws a short dimension line.



Select objects: (Press ENTER to end object selection.)

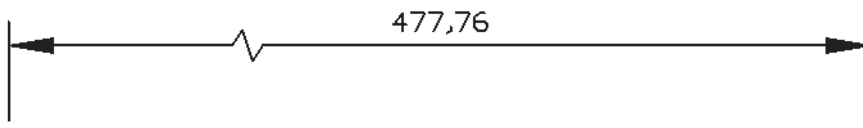
If you select dimensions instead of arrowheads, AutoCAD flips the arrowhead nearest to your pick point.



DIMJOGLINE

The DIMJOGLINE command adds jogs to linear dimension lines.

Very wide drawings, such as road cross-sections, use jogged dimension lines to show overall distance. This command adds and removes the jogs; you use the **PROPERTIES** command to specify the size of the jog, and can relocate the jog using grips editing.



The DIMJOGLINE command works only with linear dimensions. You cannot give a dimension line more than one jog; to add jogs to radial extension lines, use the DIMJOGGED command, as described in the previous chapter.

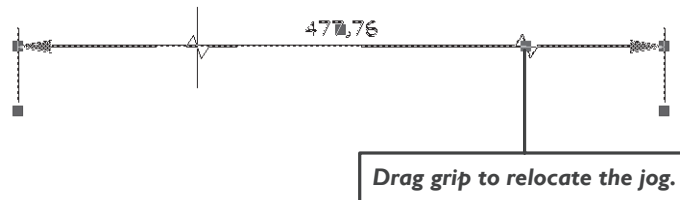
TUTORIAL: JOGGING DIMENSION LINES

- To add a jog to dimension lines, start the **DIMJOGLINE** command:
 - In the ribbon's Annotative tab, choose the **Jog Line** button in the Dimensions panel.
 - At the 'Command:' prompt, enter the **dimjogline** command.
Command: **dimjogline** (Press ENTER.)
 - Alternatively, enter the **djl** alias at the 'Command:' prompt.
- In all cases, AutoCAD prompts you to select a dimension:
Select dimension to add jog or [Remove]: (Pick one dimension.)
- Specify the location of the jog:
Specify jog location (or press ENTER): (Pick a point.)

When you press ENTER at the last prompt, AutoCAD places the jog halfway between the first extension line and the dimension text.

To change the size of the jog, use the **PROPERTIES** command: in the Lines & Arrows section, change the value of **Jog Height Factor**.

To move the jog, use grips editing (as illustrated below), or else reuse this command and pick another location for the jog. Since dimension lines can have only one jog, specifying a "second" one relocates the jog.



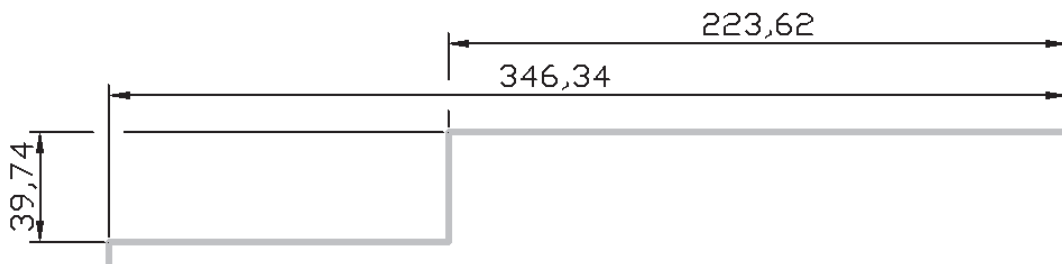
The **Remove** option removes the jog from the dimension line.



DIMBREAK

The **DIMBREAK** command breaks extension and dimension lines where they cross other objects.

Breaking lines can make dimensions easier to read. AutoCAD remembers the breaks, so that when you move a dimension, the break moves appropriately.



TUTORIAL: BREAKING DIMENSION AND EXTENSION LINES

1. To break lines in dimensions, start the **DIMBREAK** command:
 - In the ribbon's Annotative tab, choose the **Break** button in the Dimensions panel.
 - At the 'Command:' prompt, enter the **dimbreak** command.
Command: **dimbreak** (Press ENTER.)
2. In all cases, AutoCAD prompts you to select one or more dimensions:
Select a dimension or [Multiple]: (Select a dimension.)
3. Choose an option; I recommend the **Auto** option, which lets AutoCAD handle the breaking job:
Select object to break dimension or [Auto/Restore/Manual] <Auto>: **a**
Notice that dimension and/or extension lines are broken.

To place breaks manually, enter "M" for the **Manual** option, which operates like the **BREAK** command:

Specify first break point: (Pick a point.)
Specify second break point: (Pick another point.)

This option allows you to place the break anywhere, and not just where lines overlap. This option works better when osnaps are turned off.

To break all dimensions in the drawing at once, use the **Multiple** option, as follows:

Select a dimension or [Multiple]: **m**
Select dimensions: **all**
Select dimensions: (Press ENTER to exit object selection.)
Enter an option [Break/Restore] <Break>: (Press ENTER to accept the default.)

When you stretch or move dimensions, the breaks follow. Should the dimensions no longer cross, the breaks heal, but reappear when you move the dimensions back.

To remove breaks, use the **Restore** option. You can restore dimensions one by one, or all of them at once.

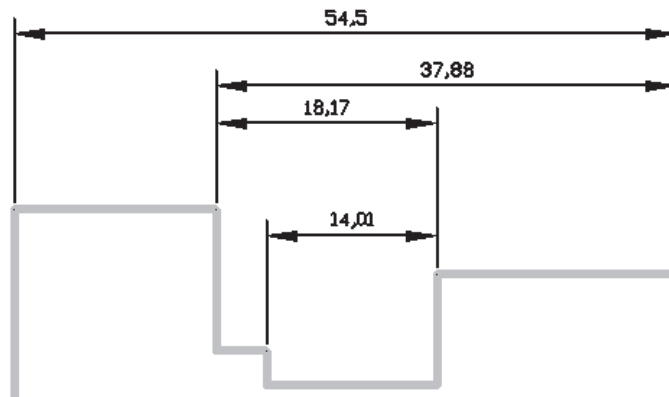


The **DIMSPACE** command evenly spaces groups of dimensions.

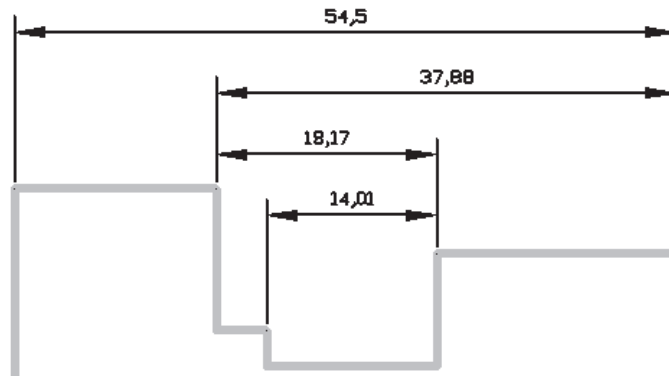
The purpose of this command is to give a neater look to drawings with many dimensions. When you stretch or move dimensions, the break follows.

TUTORIAL: EVENLY SPACING DIMENSIONS

1. To learn how to space dimensions evenly, draw several linear dimensions, as illustrated below.
2. Start the **DIMSPACE** command:
 - In the ribbon's Annotative tab, choose the **Adjust Space** button in the Dimensions panel.
 - At the 'Command:' prompt, enter the **dimspace** command.
Command: **dimspace** (Press ENTER.)

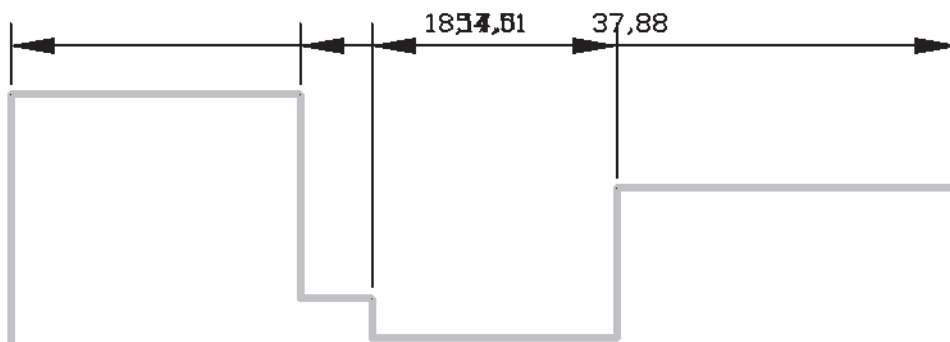


3. In all cases, AutoCAD prompts you to select the base dimension. This is the one from which the others will be spaced:
Select base dimension: *(Pick one dimension.)*
 4. Select the other dimensions to be spaced:
Select dimensions to space: *(Pick one or more dimensions.)*
Select dimensions to space: *(Press ENTER to exit object selection.)*
 5. Enter a number for the size of the spacing between dimensions; or, do it the easy way, and press **ENTER** to let AutoCAD work out the spacing:
Enter value or [Auto] <Auto>: *(Press ENTER.)*
- Notice that the dimensions have the same spacing between them.



The **Auto** option determines the spacing distance by multiplying the dimension text height; Autodesk says the factor is 2x, but in the figure above, the factor works out to 5x. This system sometimes does not work well when **DIMSCALE** is set to extreme values.

You can enter a value of 0, which reduces the spacing to zero, making the dimensions look like overlapping continuous dimensions, as illustrated below.



 DIMEDIT, **DIMTEDIT** AND **DDEDIT**

The **DIMEDIT** and **DIMTEDIT** commands edit extension lines and dimension text.

Extension lines can be *obliqued* with the **DIMEDIT** command, so that they are no longer at right angles to the dimension line. Obliqued (slanted) extension lines are required by isometric drawings, and are also useful in difficult dimensioning situations, such as dimensioning along shallow curves. Obliquing is not applied to leaders, such as diameter, radius, and ordinates, because their angles can be changed with grips editing.

The **DIMTEDIT** command changes the location of dimension text relative to the dimension line by rotating, relocating, and justifying (left, center, or right). After these changes, either command can be used to “home” the text by sending it back to its default location: centered on the dimension line.

One command’s name is short for “dimension edit,” while the other is short for “dimension text edit,” yet both edit dimension text. The toolbar icons are unhelpful, **DIMTEDIT** showing a pencil, implying the dimension itself can be edited when it cannot.

The two commands share text editing tasks, making the difference between them confusing. The table below compares and contrasts the abilities of each command:

Editing Operation	DimEdit	DimTEdit
Extension Lines:		
Apply obliquing angle	☒	...
Dimension Text:		
Rotate	☒	☒
Move to default location (home)	☒	☒
Edit text	☒	...
Drag text to new location	...	☒
Center text on dimension line	...	☒
Right- and left-justify text on dimline	...	☒

It would make more sense if **DIMTEDIT** handled *only* the editing of dimension text, while **DIMEDIT** handled the non-text parts of dimensions, such as extension lines, as well as functions ignored by AutoCAD, such as relocating and removing arrowheads. But they don’t, and so we carry on. Here’s how to keep them straight:

DimEdit obliques extension lines (ignoring its other capabilities).

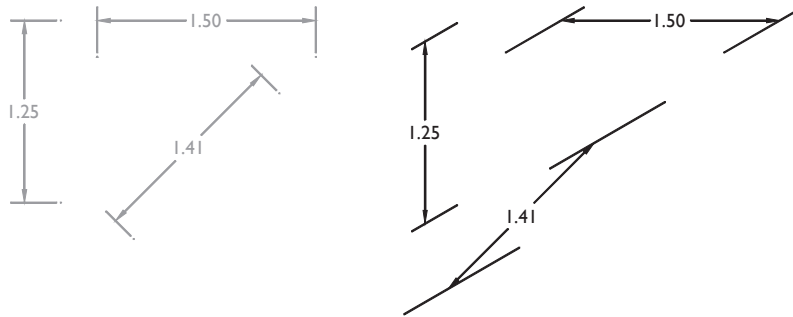
DimTEdit changes the position of dimension text.

DdEdit edits dimension text with the mtext editor.

TUTORIAL: OBLIQUING EXTENSION LINES

- To change the angle of extension lines, start the **DIMEDIT** command:
 - In the ribbon’s Annotative tab, choose the **Oblique** button in the Dimensions panel.
 - At the ‘Command:’ prompt, enter the **dimedit** command.
Command: **dimedit** (Press ENTER.)
 - Alternatively, enter the aliases **ded** or **dimed** at the ‘Command:’ prompt.
- In all cases, AutoCAD lists the options available. Enter **o** for the oblique option:
Enter type of dimension editing [Home/New/Rotate/Oblique] <Home>: **o**

3. Select the dimensions whose extension lines to slant.
 You can enter **all**, and AutoCAD filters out non-dimension objects.
 Select objects: *(Pick one or more dimensions.)*
 Select objects: *(Press ENTER to end object selection.)*
4. Specify the angle of slant between 0 and 360 degrees:
 - To slant extension lines for isometric drawings, enter **30**.
 Enter obliquing angle (press ENTER for none): **30**
 - To slant in the other direction, enter negative angles, such as **-30**.
 - To keep the extension lines at their current angle (no change), press **ENTER**.
 - To return obliqued extension lines to perpendicular, enter **0** or **90**.
 - Or, pick two points to show AutoCAD the angle.
 AutoCAD obliques the extension lines of selected dimensions.



Left: Dimensions with normal, perpendicular extension lines.
Right: Dimensions with extension lines obliques by 30 degrees.

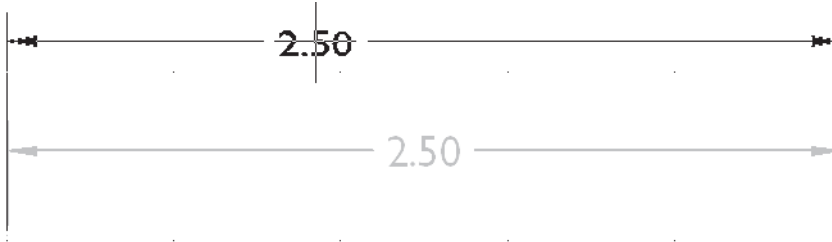


Note Be careful with obliquing angles that are very close to the angle of the dimension (1.0 degrees for horizontal dimensions, 89.0 degrees for vertical dimensions, and so on). They stretch the dimension into an unrecognizable jagged line. You can always use the **U** command to reverse undesirable obliquing.

REPOSITIONING DIMENSION TEXT

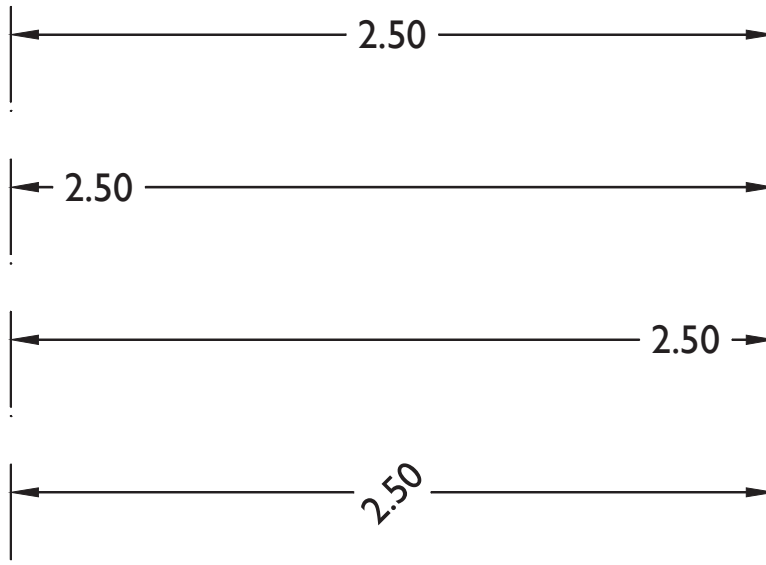
1. To change the position of dimension text, start the **DIMTEDIT** command:
 - In the ribbon's Annotative tab, choose one of the **Justify** buttons in the Dimensions panel.
 - At the 'Command:' prompt, enter the **dimtedit** command.
 Command: **dimtedit** *(Press ENTER.)*
 - Alternatively, enter the **dimted** alias at the 'Command:' prompt.
2. AutoCAD prompts you to select one dimension:
 Select dimension: *(Select one dimension.)*
3. Enter an option to relocate the text:
 Specify new location for dimension text or [Left/Right/Center/Home/Angle]: *(Move cursor, or enter an option.)*

- **Specify new location** relocates the text along the dimension line dynamically, and at the same time moves the dimension line perpendicular to the extension lines.



- **Left** moves the text to the left end of the dimension line.
- **Right** moves the text to the right end of the dimension line.
- **Center** centers the text on the dimension line.
- **Home** returns the text to its original position.
- **Angle** rotates text about its center point. AutoCAD prompts you:
Specify angle for dimension text: *(Enter an angle, such as 30.)*

Angled text is used with isometric dimensions.



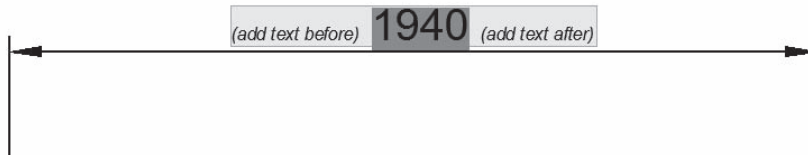
Top: Dimension text in home and center positions.

Above: Text in left, right, and rotated positions.

EDITING DIMENSION TEXT

- To edit dimension text, start the **DDEDIT** command:
 - At the 'Command:' prompt, enter the **ddedit** command.
Command: **ddedit** (Press ENTER.)
 - Alternatively, enter the **ed** alias at the 'Command:' prompt.
- In all cases, AutoCAD prompts you to select the text to edit.
Select a single dimension:
Select an annotation object or [Undo]: *(Select a dimension or leader text.)*

3. AutoCAD opens the mtext editor with the dimension text highlighted in blue.



Recall from the previous chapter that you can edit the dimension text (with the blue background):

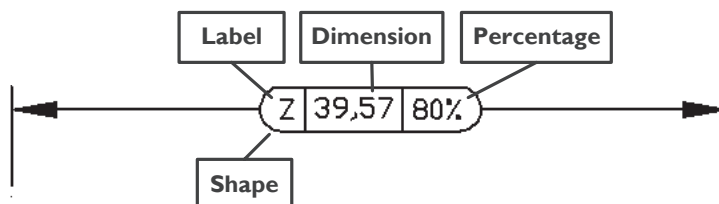
- **Erase** the text from the dimension.
 - **Change** the dimension text.
 - **Add** text before and after to create prefixes and suffixes.
4. When done editing the text, click **OK** on the Text Formatting bar.
 5. AutoCAD repeats the prompt.
Select another dimension, or any other text object.
When done editing, press **ENTER**.
Select an annotation object or [Undo]: (Press **ENTER** to exit command.)

To apply a background mask to the dimension text, use the **Fill** option of the Properties palette.

DIMINSPECT

The DIMINSPECT command adds and edits inspection-style text to dimensions.

Inspection dimensions are used in quality assurance (QA) testing to ensure that the manufactured part has precisely the correct dimensions. For example, if the percentage is 80%, then 80% of the manufactured parts must meet this dimension to pass QA. This command modifies existing dimension text to make three parts:

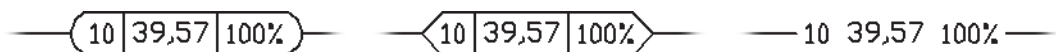


Label — identifies the dimension in tables and notes. You can enter any text for the label.

Dimension — specifies the original dimension text. Like other dimensions, inspection dimensions can contain prefixes, suffixes, and tolerances.

Percentage — specifies the inspection rate. You can enter any value, such as 23/74 or “Quite a few”; it need not be a percentage.

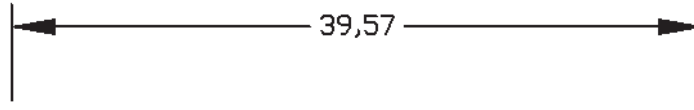
Shape — surrounds the dimension to make it stand out. AutoCAD allows ovals, diamonds, or no shape at all.



You select the type and value of inspection through a dialog box; the **-DIMINSPECT** command performs the same function at the command line.

TUTORIAL: ADDING INSPECTION TEXT TO DIMENSIONS

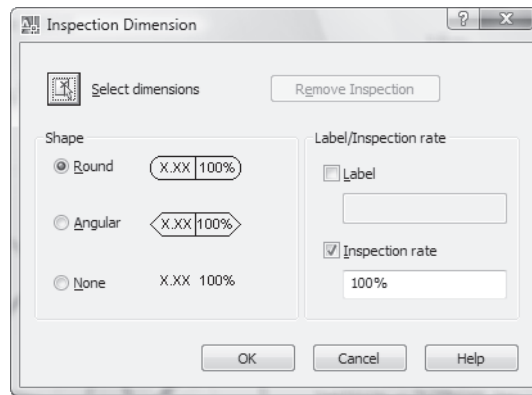
1. To add inspection text to dimensions, draw a linear dimension similar to the one illustrated below.



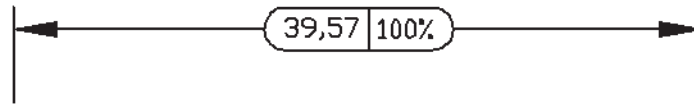
2. To add inspection text, start the **DIMINSPECT** command:
 - From the Dimension menu, choose **Inspection**.
 - From the Dimension toolbar, choose the **Inspection** button.
 - In the ribbon's Annotative tab, choose the **Diminspect** button from the Dimensions panel.
 - At the 'Command:' prompt, enter the **diminspect** command.

Command: **diminspect** (Press ENTER.)

3. In all cases, AutoCAD displays the Inspection Dimension dialog box:



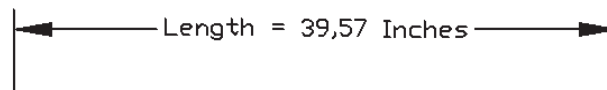
4. Click the **Select Dimensions** button.
5. Notice that AutoCAD prompts you at the command line:
 Select dimensions: (Select one or more dimensions.)
 Select dimensions: (Press ENTER to return to the dialog box.)
6. Choose inspection options, and then click **OK**. The default options change the dimension to this:



To change the values and/or shape of inspection dimensions, rerun the **DIMINSPECT** command, and then select the dimension. Alternatively, you can use the Misc section of the Properties palette to edit inspection dimensions.



Note You can use this command to create dimensions interactively with prefixes and suffixes, such as the one illustrated below. Set **Inspection Label** to "Length = " and **Inspection Rate** to "Inches." Alternatively, you can use the Properties palette to add and edit prefixes and suffixes.



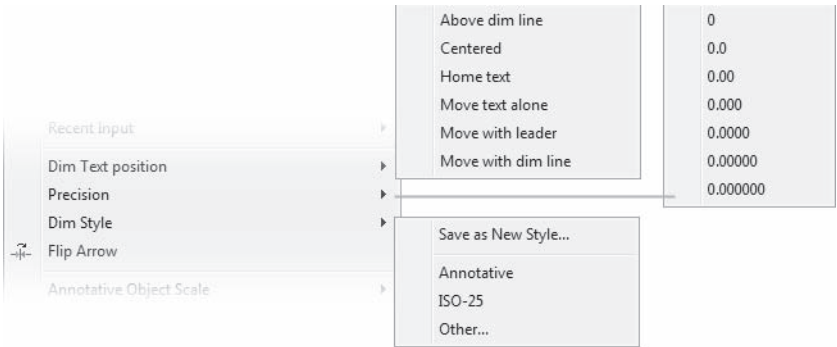
AIDIM AND AI_DIM

AutoCAD has a group of undocumented commands that edit dimension text. These commands are meant for use in menu and toolbar macros, but are also handy for making quick fixes to mucked-up dimensions. The commands are shortcuts to options within other commands and dimension variables:

(The “AI” prefix indicates custom commands written by Autodesk Incorporated. Why do some commands start with AI and some with AI_? It seems that those starting with AI have multiple options, while those with AI_ perform a single operation.)

Command	Shortcut For	Comment
AIDIMPREC	DIMDEC	Changes precision of fractions and decimals selectively.
AIDIMSTYLE	DIMSTYLE	Saves and applies up to six dimension styles.
AIDIMTEXTMOVE	DIMTMOVE	Relocates text with optional leader.
AI_DIM_TEXTABOVE	DIMTAD 3	Moves text above dimension line for JIS compliance.
AI_DIM_TEXTCENTER	DIMTEDIT C	Centers dimension text.
AI_DIM_TEXTHOME	DIMTEDIT H	Returns text to home position.

You find these commands in a shortcut menu: select a dimension, and then right-click.



Here is how the commands relate to the shortcut menu:

Shortcut Menu	Submenu	Related Command	Option
Dim Text position	► Move text alone	AIDIMTEXTMOVE	0
	► Move with leader	AIDDIMTEXTMOVE	1
	► Move with dim line	AIDIMTEXTMOVE	2
	► Above dim line	AI_DIM_TEXTABOVE	...
	► Centered	AI_DIM_TEXTCENTER	...
	► Home text	AI_DIM_TEXTHOME	...
Precision	...	AIDIMPREC	0 – 6
Dim Style	► Save as New Style	AIDIMSTYLE	Save
	► Standard, etc.	AIDIMSTYLE	1 – 6
	► Other	AIDIMSTYLE	Other
Flip Arrow	...	AIDIMFLIPARROW	

AIDIMPREC

The AIDIMPREC command selectively changes the precision of fractional and decimal dimension text and angles of all dimensions type, except leaders. To affect a permanent change, change the dimstyle.

This command is more convenient than its official alternative: changing the **DIMDEC** system variable, followed by the **-DIMSTYLE** command's **Apply** option. Decimal text and angles are rounded to the nearest decimal place, while fractional text is rounded to the nearest fraction.

AiDimPrec	Decimal Units	Fractional Units
0	0	1"
1	0.0	1/2"
2	0.00	1/4"
3	0.000	1/8"
4	0.0000	1/16"
5	0.00000	1/32"
6	0.000000	1/64"

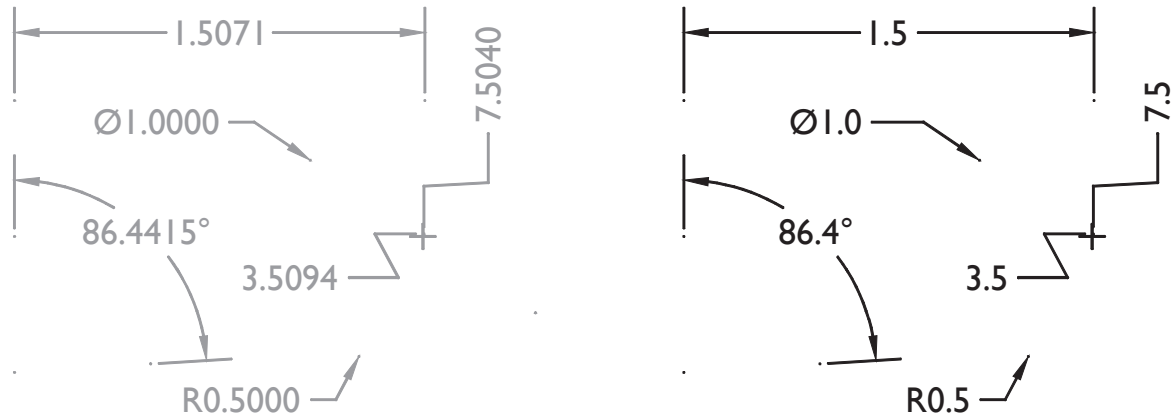
CHANGING DISPLAY PRECISION

- 1. To change the precision of dimension text, start the **AIDIMPREC** command:
Command: **aidimprec** (Press **ENTER**.)
- 2. AutoCAD prompts you to specify a precision:
Enter option [0/1/2/3/4/5/6] <4>: (Enter the number of decimal places.)
- 3. Select one or more dimension objects.
Select objects: (Enter **all** or pick one or more dimensions.)
Select objects: (Press **ENTER** to end object selection.)

Entering **all** selects everything in the drawing; AutoCAD filters out non-dimension objects.

This command does not change the precision of leader objects.

Notice that AutoCAD changes the display precision of the selected dimensions.



Left: Dimension text displayed to four decimal places.
Right: Precision reduced to one decimal place.



Notes The change in precision is not permanent; AutoCAD remembers the actual measurements and angles — which is why it is called “display precision.” You can reapply the command to increase and decrease the precision displayed by the dimensions.

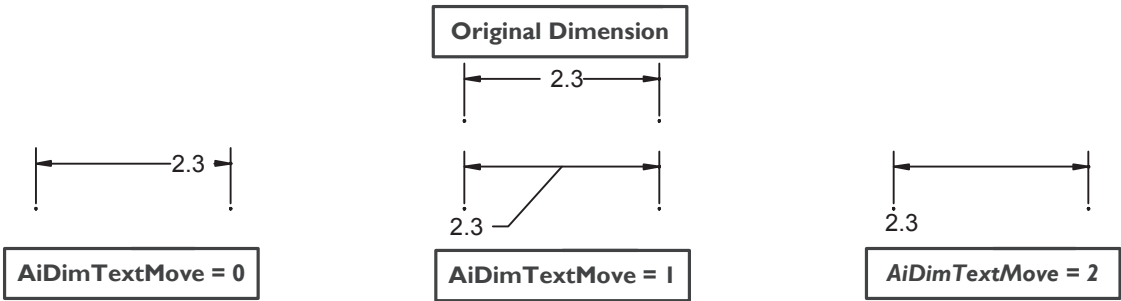
Because the **AIDIMPREC** command rounds off dimension text, it can create false measurements. For instance, if a dimension measures 3.4375", setting **AIDIMPREC** to 0 rounds down to 3". Similarly, a measurement of 2.5" rounds up to 3".

 AIDIMTEXTMOVE

The AIDIMTEXTMOVE command relocates text, either with or without moving the dimension line, and optionally adds a leader.

This command is more convenient than its official alternative: changing the DIMTMOVE system variable, followed by the DIMOVERRIDE command. AIDIMTEXTMOVE has the same options as DIMTMOVE:

AiDimTextMove	DimTMove	Comment
0	0	Moves text within dimension line; moves dimension line.
1	1	Adds a leader to the moved text.
2 (Default)	2	Moves text without leader line; moves the dimension line.



MOVING DIMENSION TEXT

1. To move the dimension text, start the **AIDIMTEXTMOVE** command:
Command: **aidimtextmove** (Press ENTER.)
2. AutoCAD prompts you to specify a precision:
Enter option [0/1/2] <2>: (Enter an option.)
3. Select one dimension.
Although the command allows you to select more than one dimension, it operates on the first-selected dimension only.
Select objects: (Pick one dimension.)
Select objects: (Press ENTER to end object selection.)
4. Move the cursor to relocate the dimension text.

The following trio of commands quickly relocates dimension text relative to the dimension line: above, centered, or in the dimension line. These commands operate on just one dimension at a time, unfortunately. Using this command is more convenient than changing the DIMTAD system variable, followed by the -DIMSTYLE command's Apply option.

Command	DimTAD	Comment
AI_DIM_TEXTCENTER	0	Centers text vertically.
AI_DIM_TEXTHOME	2	Centers text horizontally and vertically.
AI_DIM_TEXTABOVE	3	Moves text above dimension line.

(When DIMTAD is 1, AutoCAD moves text above the dimension line — when the dimension line is horizontal, and text inside the extension lines isn't forced horizontally.)

Ai_Dim_TextCenter

The `AI_DIM_TEXTCENTER` command centers text along the dimension line. If the text is above or below the dimension line, AutoCAD keeps the vertical position, but centers the text horizontally.

Command: **ai_dim_textcenter**

Select objects: *(Pick one dimension.)*

Select objects: *(Press ENTER to end object selection.)*

Ai_Dim_TextHome

The `AI_DIM_TEXTHOME` command centers the dimension text on the side of the dimension line farthest from the defining points.

Command: **ai_dim_texthome**

Select objects: *(Pick one dimension.)*

Select objects: *(Press ENTER to end object selection.)*

Ai_Dim_TextAbove

The `AI_DIM_TEXTABOVE` command moves text above the dimension line, which is helpful for making drawings JIS compliant.

Command: **ai_dim_textabove**

Select objects: *(Pick one dimension.)*

Select objects: *(Press ENTER to end object selection.)*

DIMENSION VARIABLES AND STYLES

How dimensions look and act depends on the settings of *dimension variables* (called “dimvars” for short). The variables control many properties of dimensions, such as the placement of dimension text within or above the dimension line, the use of arrowheads or tick marks, the color of the extension lines, and the overall scale. (Technically, dimvars are no different from system variables.)

The names of all dimvars begin with “dim,” which makes them easy to identify, such as `DIMSCALE` for dimension scale and `DIMTXT` for the text style. Other names, however, can seem strange and are hard to decode. Who could guess that `DIMTOFL` means “draw the dimension line between the extension lines, even when the text is forced outside due to lack of space.” I suppose “tofl” could be short for “text outside forced line.” Or how about `DIMTAD` — “Text Addjusted relative to the Dimension line,” perhaps? (The technical editor suggests a new dimension variable, `DIMWIT` for “Weird Indecipherable Terminology.”)

Like system variables, dimvars can be toggled (turned on and off), hold numeric values, or report on the status of dimensions. The advantage to dimvars is that they control almost every aspect of dimensions; the disadvantage is that there are so many of them (around 80) that it becomes difficult to remember them all; two-thirds alone are dedicated to formatting dimension text.

That’s why AutoCAD has *dimensions styles*. Like text styles, these remember dimvar settings and easily control the look of dimensions. When you create or change a dimensions style with the `DIMSTYLE` command, AutoCAD changes the values of dimvars.

Creating dimension styles (“dimstyles” for short) is similar to creating text styles: you specify options, and then save them by name. All new AutoCAD drawings based on the *acad.dwt* template drawing have two dimension styles, named “Standard” and “Annotative.”

CONTROLLING DIMVARS

AutoCAD's dimension variables can be set through several methods, such as by using the Dimension Style Manager dialog box or the Properties palette, or by entering dimvar names at the 'Command:' prompt.

For example, to set an overall scale factor for dimensions, it can be faster to enter the **DIMSCALE** at the command prompt than to hunt it down in the multi-tabbed Dimension Style Manager dialog box:

Command: **dimscale**

New value for DIMSCALE <1.0>: (Enter a scale factor, such as **2.5**.)

The scale is not retroactive, however, and only applies to dimensions drawn subsequently. To change the scale of existing dimensions, you need to use the **DIMSTYLE** dialog box's **Override** option.

There are two dimvars not set by the Dimension Style Manager dialog box: **DIMSHO** (toggles whether dimensions are updated while dragged) and **DIMASO** (determines whether dimensions are associative or non-associative). Both have a value of 1, which means they are turned on. You can turn them off, but there isn't any good reason to do so.

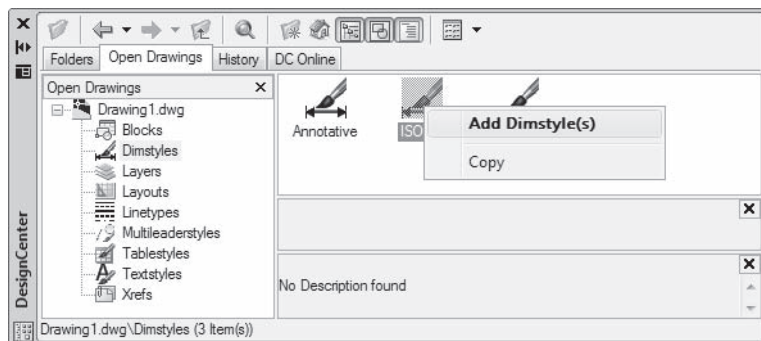
(History: Both of these dimvars exist only for compatibility with old versions of AutoCAD. When computers were slow, generating highlighted images of dragged dimensions slowed the computer even more; this is no longer an issue.)

SOURCES OF DIMSTYLES

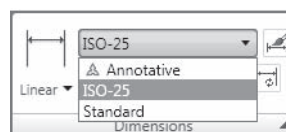
When you create new drawings based on template drawings, they contain specific dimstyles. AutoCAD includes a *.dwt* template file that meets the international metric standard: *acad_iso.dwg* contains a dimstyle named "ISO," which conforms to the dimensioning standards of the International Organization for Standardization.

Dimstyles from Other Sources

To copy dimension styles from other drawings, use DesignCenter. In the tree view of DesignCenter, open the **Dimstyles** item in the drawing from which you wish to copy the dimstyles. Right-click the dimstyle name, and then select **Add Dimstyle(s)** from the cursor menu.

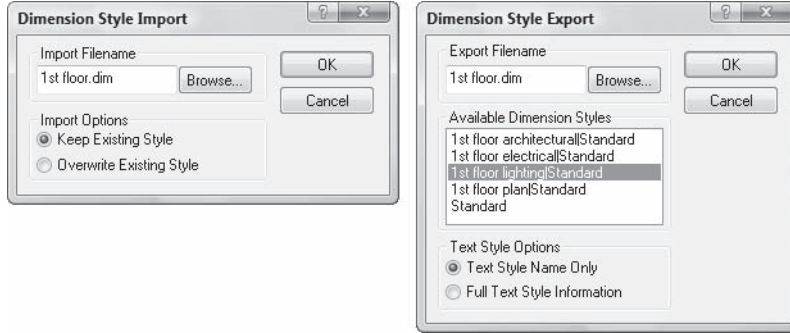


To check that the dimstyle was added to the drawing, click the **Dimstyle Control** list box on the Styles toolbar or ribbons's Dimensions panel. The name should appear on the list.

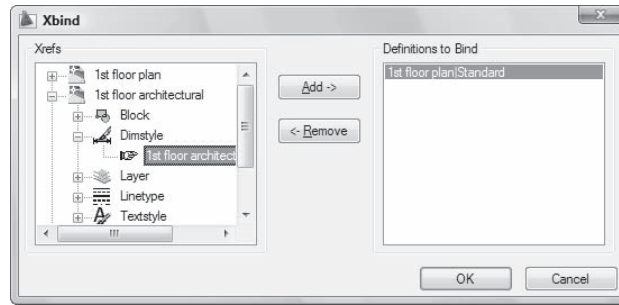


AutoCAD has several additional methods of sharing dimension standards between drawings:

Express Tools includes the **DIMEX** and **DIMIM** commands for exporting and importing files that contain dimension styles. From the **Express Tools** tab, click the **DimStyle Export Style** or **Import Style** button on the Dimension panel. AutoCAD displays the dialog boxes shown below.



When working with externally-referenced drawings, you can use the **XBIND** command to *bind* (add in) dimension styles found in attached drawings. You cannot, however, set an externally-referenced dimstyle as current.



See appendix D, "Summary of Dimension Variables," for the complete list of dimension variables.

DIMSTYLE

The **DIMSTYLE** command displays a dialog box for creating and modifying named dimension styles.

The tabbed dialog boxes of the Dimension Style Manager control almost all dimension variables, which in turn affect the look of dimensions in your drawings. A preview window provides a graphical overview of the current dimstyle.

New drawings contains at least two dimension styles, either "Standard" in Imperial drawings or "ISO-25" in metric drawings, and "Annotative" in all drawings. You can create as many dimstyles as you require, each with its own name — one for every drawing, if need be.

The dialog box is called a "manager," because it manages dimension styles in the drawing. With it, you create new styles, override and modify existing styles, compare differences between two styles, and delete styles.

WORKING WITH DIMENSION STYLES

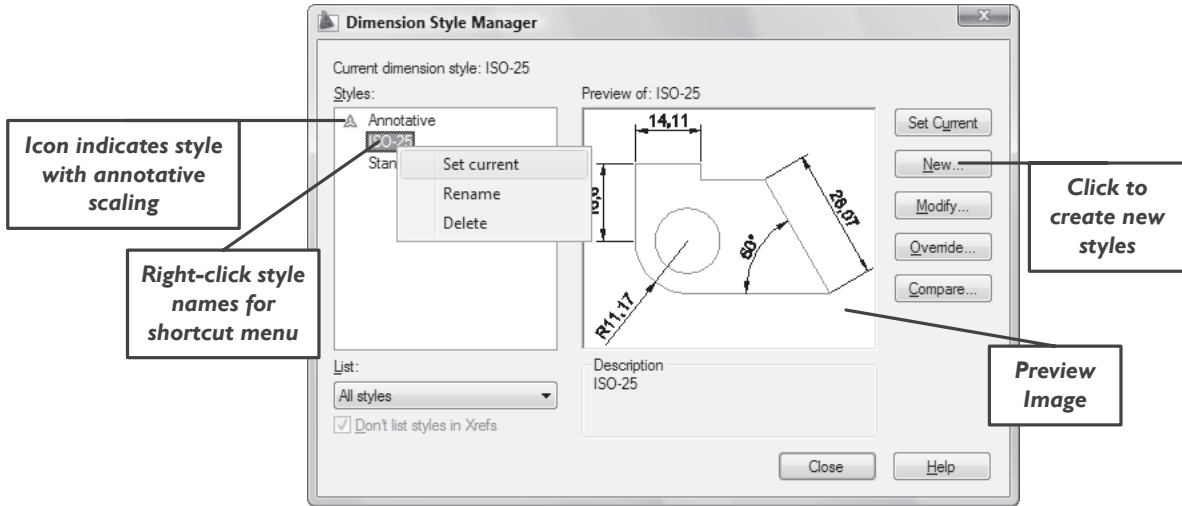
To create and change dimension styles, start the **DIMSTYLE** command:

- In the ribbon's **Annotative** tab, choose the **Dimension Style** button in the Dimensions panel.
- At the 'Command:' prompt, enter the **dimstyle** command.

Command: **dimstyle** (Press ENTER.)

- Alternatively, enter the aliases **d**, **dst**, **dimsty**, or **ddim** (the command's old name) at the 'Command:' prompt.

In all cases, AutoCAD displays the Dimension Style Manager dialog box.

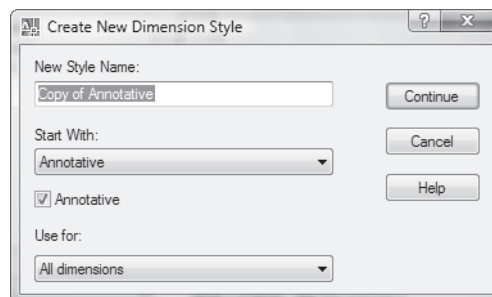


On the right, five buttons perform these important functions, and are discussed in detail below.

- Set Current** — sets the selected dimstyle current. After clicking this button, you usually click **OK** to exit the dialog box. After you exit, new dimensions follow this style.
- New** — creates new dimstyles by naming them; usually works by renaming and modifying an existing style, such as Standard or Annotative.
- Modify** — modifies retroactively, and globally updates, all existing dimensions created using the style. You tend to use this button more often than **Override**.
- Override** — overrides the settings of the current dimstyle. Overrides change only dimensions created after the override is applied. Externally-referenced dimstyles cannot be modified or overridden.
- Compare** — displays a dialog box for contrasting the differences between two dimension styles.

NEW — CREATING NEW DIMSTYLES

- To create a new dimension style, click the **New** button, which leads to the Create New Dimension Style dialog box.
- Making a new style is as easy as typing a name of up to 255 characters long.



- Choose **Continue**. Notice that AutoCAD displays the New Dimension Style dialog box, which is identical to the Modify version of the dialog box detailed below.



Notes The standard method of creating a new style is to copy an existing one, make changes, and then save the result. For this reason, AutoCAD displays “Copy of Standard” (or whatever the current style name is) in the **New Style Name** text field.

Start With text field lets you select which dimstyle to copy — provided the drawing contains two or more dimstyles. This allows you to start with an existing dimstyle (such as the DIM, JIS, Gb, and ISO styles stored in the template drawings) and modify it according to your needs.

Annotative determines whether the style uses annotative scaling.

Use for droplist lets you apply changes to all, or a limited group of, dimensions:

- All dimensions
- Angular dimensions
- Radius dimensions
- Leaders and tolerance
- Linear dimensions
- Ordinate dimensions
- Diameter dimensions

MODIFY — MODIFYING DIMSTYLES

The **Modify** button leads to the Modify Dimension Style dialog box, which contains most of the options that affect dimensions. Here you modify the values to create and change dimstyles. Because there are so-o-o many options affecting dimensions, the dialog box segregates them into a series of tabs. Here is an overview of what each tab contains:

- **Lines** — properties of dimension and extension lines.
- **Symbols and Arrows** — properties of arrowheads, sizes of center marks and arc length symbols, and angle of jogs.
- **Text** — format, placement, and alignment of dimension text.
- **Fit** — placement of dimension text, overall scaling of dimensions, and toggling of annotation.
- **Primary Units** — format of units for primary linear and angular dimensions, and zero suppression.
- **Alternate Units** — format of alternate (secondary) units.
- **Tolerances** — format of dimension tolerance text.

LINES TAB

The Lines tab specifies the properties of dimension lines and extension lines.

The screenshot shows the 'Lines' tab of the 'Modify Dimension Style' dialog box. The dialog has several tabs: 'Lines', 'Symbols and Arrows', 'Text', 'Fit', 'Primary Units', 'Alternate Units', and 'Tolerances'. The 'Lines' tab is active.

Dimension lines

- Color: ByBlock
- Linetype: ByBlock
- Lineweight: ByBlock
- Extend beyond ticks: 0.0000
- Baseline spacing: 6.0000
- Suppress: ☐ Dim line 1 ☐ Dim line 2

Extension lines

- Color: ByBlock
- Linetype ext line 1: ByBlock
- Linetype ext line 2: ByBlock
- Lineweight: ByBlock
- Suppress: ☐ Ext line 1 ☐ Ext line 2
- Extend beyond dim lines: 3.0000
- Offset from origin: 1.5000
- ☐ Fixed length extension lines
- Length: 1.0000

On the right side of the dialog, there is a preview window showing a technical drawing of a part with various dimensions. The dimensions include a horizontal distance of 1.02, a vertical distance of 1.20, a radius of R0.80, and an angle of 60°. The dimension lines and extension lines are shown with the settings specified in the dialog.

Dimension Lines

The Dimension Lines section specifies the properties of dimension lines.

Color

The **Color** droplist specifies the color of dimension lines. The default color is Byblock, because dimensions are considered blocks. “ByBlock” means that dimension lines take on the color of the entire dimension; in contrast, the dimension’s default color is Bylayer, meaning the dimension takes on the color defined by its layer.

The color of the dimension line is stored in the dimension variable named **DIMCLRD** (short for “DIMension CoLoR Dimension line”).



Note Although you can choose another color, it is best to place dimensions on layers that specify the color of the dimension.

You can see the effect of changing variables through the Preview box: select the color blue from the **Color** droplist. Notice in the Preview box that the dimension line color changes from black to blue.

Linetype

To assign linetypes to dimension lines, click the **Linetype** droplist (stored in **DIMLTYPE**). The preview window updates to show the new linetype.

You cannot change the linetype if *any* dimstyle override is active. As with colors, it is better to specify the lineweight via the layer, rather than to change the lineweight here.

Lineweight

To change the lineweight of the dimension line, choose the droplist next to Lineweight and select a predefined width. The dimension line lineweight is stored in dimension variable **DIMLWD**.

As with colors and linetypes, it is better to specify the lineweight via the layer, than to change the lineweight here.

Extend Beyond Ticks

The **Extend beyond ticks** option sets the distance that extension lines extend beyond the dimension line (stored in **DIMDLE**). The default is 0.0.

This option is available only when the “Architectural Tick” is selected for arrowheads, an option that’s set in the Symbols and Arrows tab.

Baseline Spacing

The **Baseline spacing** option determines the distance between dimension lines when they are automatically stacked by the **DIMBASELINE** and **QDIM** commands.

The distance is stored in dimvar **DIMDLI**.

Suppress

AutoCAD can suppress the display of either or both dimension lines. To do so, select the **Suppress** check boxes to suppress **Dim Line 1** (stored in dimvar **DIMSD1**) or **Dim Line 2** (dimvar **DIMSD2**). These options also suppress the display of associated arrowheads.

Dim Line 1 refers to neither the left nor right part of the dimension line. Instead, it is the first dimension line drawn, as determined by the initial pick points that place the extension lines.

When the text does not cut the dimension line in two, these options do not affect the dimension line.

Extension Lines

The **Extension Lines** section specifies the properties of extension lines.

Color, Linetype, and Lineweight

The **Color**, **Linetype**, and **Lineweight** options are similar to those of dimension lines; you can specify the color (stored in dimvar DIMCLRE) and lineweight (DIMLWE) of extension lines. In the case of linetypes, you can specify a different one for each extension line (DIMLTEx1 and DIMLTEx2).

Suppress

The **Suppress** option hides the first (DIMSE1) or second (DIMSE2), or both extension lines. “Ext line 1” is the first extension line drawn, based on the points picked during the dimensioning command.

(History: this was an issue for pen plotters. When multiple extension lines were drawn on top of each other with pens, the paper eventually wore through.)

Additional Options

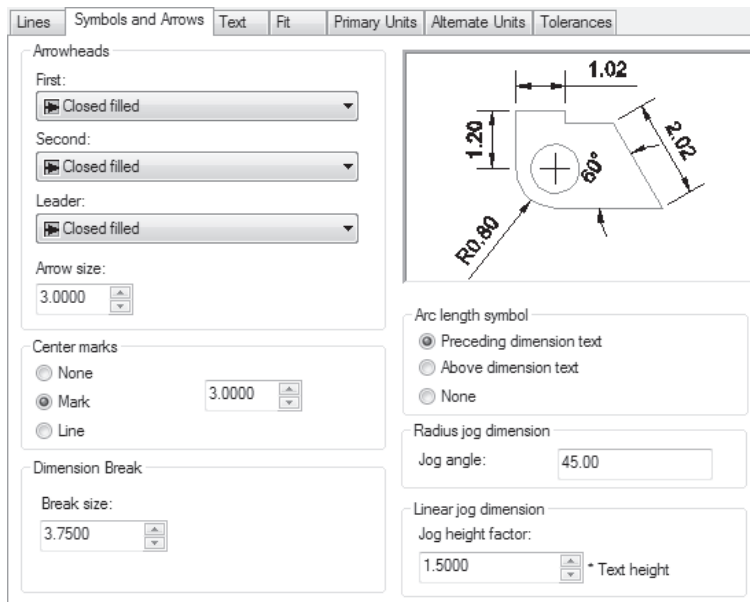
The **Extend beyond dim lines** option specifies the distance the extension line protrudes beyond the dimension line (stored in DIMEXE).

The **Offset from origin** option sets the distance the extension line begins away from the pick point of the object being dimensioned (DIMEXO).

The **Fixed length extension lines** option specifies a fixed length for all extension lines (DIMFXLON). The default value is 0.18 units (DIMFXL).

SYMBOLS AND ARROWS

The Symbols and Arrows tab specifies the properties of extension lines.



Arrowheads

The Arrowheads section defines the type and size of arrowheads used for dimensions and leaders.

First, Second, and Leader

The **First** droplist lets you choose from 20 predefined arrowhead styles (stored in DIMBLK). In addition to the standard arrowhead, AutoCAD includes arrowheads, such as the tick, the dot, open arrowhead, open dot, right angle arrowhead, or no head at all; see figure at the side.

When you select an arrowhead from the **First** droplist, AutoCAD changes the Second and Leader droplists to the same arrowhead.

Alternatively, you can select **User Arrow** to choose a custom-made arrowhead. The customized arrowhead is defined by the creation of an object scaled to unit size, and then saved as a named block with the **BLOCK** command. The arrowhead name is stored in DIMBLK. (See the tutorial on the following pages.)

Second and Leader

You can specify a different arrowhead for each end of dimension lines (DIMBLK1 and DIMBLK2) and for the leader dimensions (DIMLDRBLK). When you specify a different one for each end, AutoCAD changes dimvar DIMSAH to 1.

Although radius and diameter dimensions consist of leaders, they use the arrowhead defined by Second; the arrowhead defined by **Leader** affects only the leaders placed by the **LEADER** and **QLEADER** commands. Ordinate dimensions do not use arrowheads.

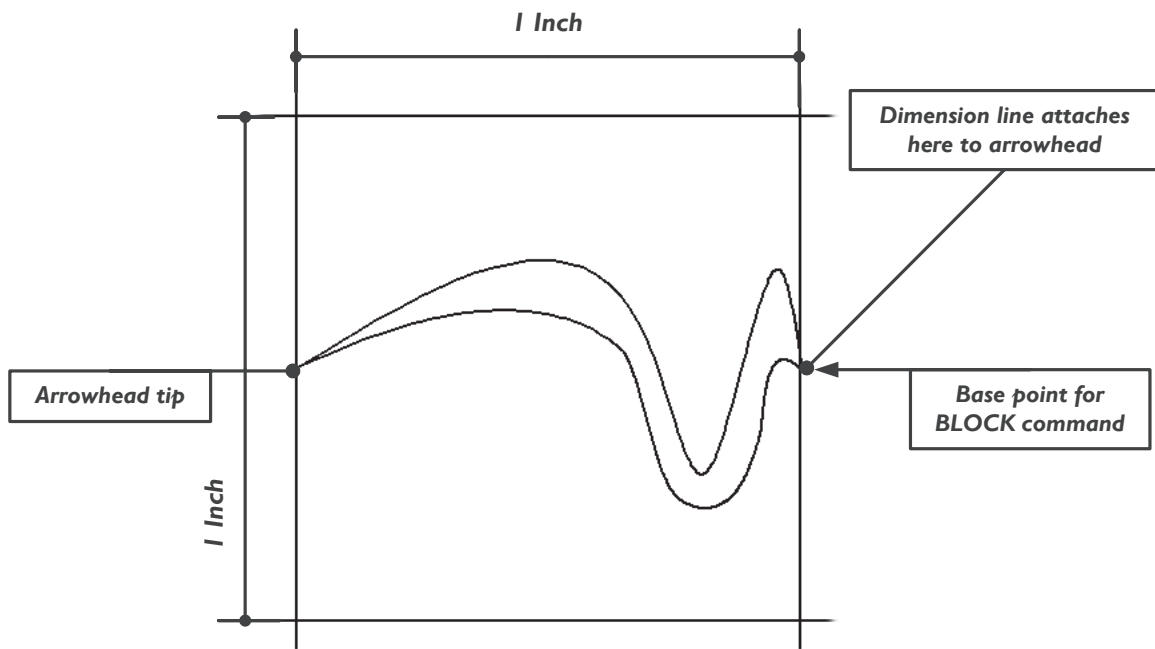
Arrow Size

The **Arrow size** option specifies the distance from the left to right end; for custom arrowheads, AutoCAD scales the unit block to size (0.18 units, by default). The value is stored in DIMASZ.

TUTORIAL: CREATING CUSTOM ARROWHEADS

In this tutorial, you draw a custom arrowhead, and then use it with dimensions.

1. Start AutoCAD with a new drawing.
2. Set the grid to 1.0, or use xlines to demarcate a one inch square, because the arrowhead must fit within a 1"-square.



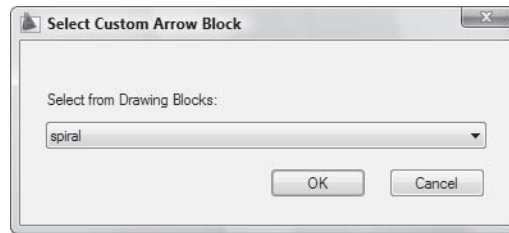
3. Draw the arrowhead with the tip pointing west.
(The arrowhead illustrated below was drawn with the **SPLINE** and **OFFSET** commands.)
The arrowhead attaches to the dimension line on the East (see figure below).

- Closed filled
- ▷ Closed blank
- ≡ Closed
- Dot
- ↗ Architectural tick
- ↘ Oblique
- ⇒ Open
- ⊙ Origin indicator
- ⊙ Origin indicator 2
- Right angle
- ⇒ Open 30
- ◆ Dot small
- Dot blank
- ◇ Dot small blank
- Box
- Box filled
- ◁ Datum triangle
- ◀ Datum triangle filled
- ↗ Integral
- None

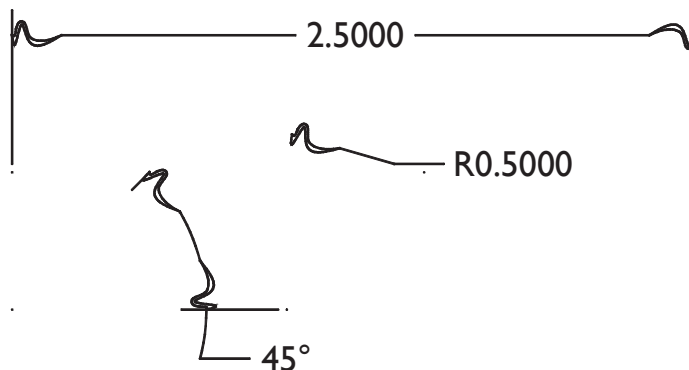
4. Use the **BLOCK** command to turn the arrowhead into a block, and then give the block a meaningful name, such as “spiral.”

Important! For the Base Point, pick the East end of the arrowhead, where it attaches to the dimension line, as illustrated below.

5. To use the new arrowhead with dimensions, do the following:
 - a. Start the **DIMSTYLE** command, and then click **Modify**.
 - b. Select the **Arrows and Symbols** tab.
 - c. Under Arrowheads, in the First droplist, select **User Arrow**. Notice the Select Custom Arrow Block dialog box, which lists the names of all blocks defined in the drawing. (You can use any block found in the drawing as an arrowhead. Because they were not designed for use as arrowheads, however, they would probably not work well.)



6. Select the name of the block you created earlier, and then click **OK**.
Notice that AutoCAD automatically fills in the **Second** and **Leader** droplists with the same name. The preview image shows the custom arrowheads.
7. Click **OK** to exit the Symbols and Arrows tab.
Choose the **Set Current** button, and then **Close** to close the dialog box.
8. Place some dimensions with the custom arrowhead.



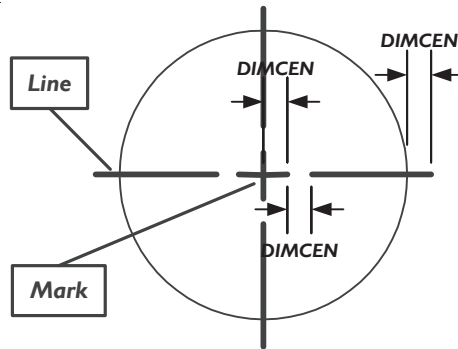
If the arrowhead is too small, increase its size with the **DIMSTYLE** command's **Arrow size** option.

Center Marks for Circles

When you mark the centers of circles and arcs with the **DIMCENTER** command, AutoCAD determines the type of center mark by the value stored in the **DIMCEN** dimvar: center mark, center mark with extending lines, or no mark at all. Center marks are also placed by the **DIMRADIUS** and **DIMDIAMETER** commands.

The value of **DIMCEN** determines the length of the marks, the width of the gap, and the distance the lines extend beyond the circle. The default value is 0.09 units. When **DIMCEN** is

- 0.09 Center lines and center marks are drawn.
- 0 No center mark or lines drawn.
- 0.09 Center marks drawn.



In the Dimension Styles dialog box, select a radio button:

- **None** places no center marks in circles and arcs.
- **Mark** places the center mark.
- **Line** places the center mark and lines.

Size specifies the size of the mark, gap, and line extension.

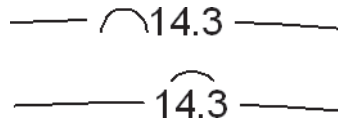
Dimension Break

Break Size specifies the size of the gap created by the DIMBREAK command.

Arc Length Symbol

The **Arc Length Symbol** option specifies the location of the arc length symbol (DIMARCSYM). It looks like an upside down “U”; the symbol cannot be customized:

- **Preceding Dimension Text** places the symbol before the dimension text.

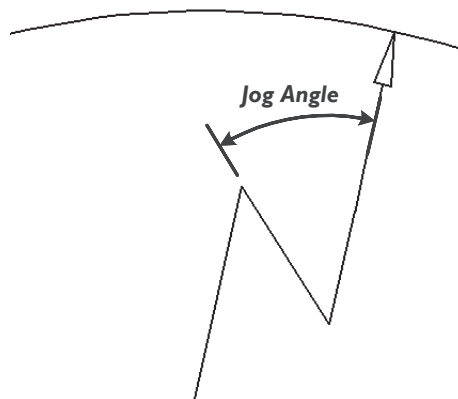


- **Above Dimension Text** places the symbol above the dimension text.
- **None** suppresses the display of the symbol.

Radius Dimension Jog

The **Radius Dimension Jog** section specifies the angle used by jogged dimensions (DIMJOGANG). The default is 45 degrees.

Use the **Jog Angle** option to change the angle. Autodesk does not document the limits, but the angle must be between 5 and 90 degrees. Enter an angle outside those limits, and AutoCAD does not allow you to exit the tab — until you enter a correct value.

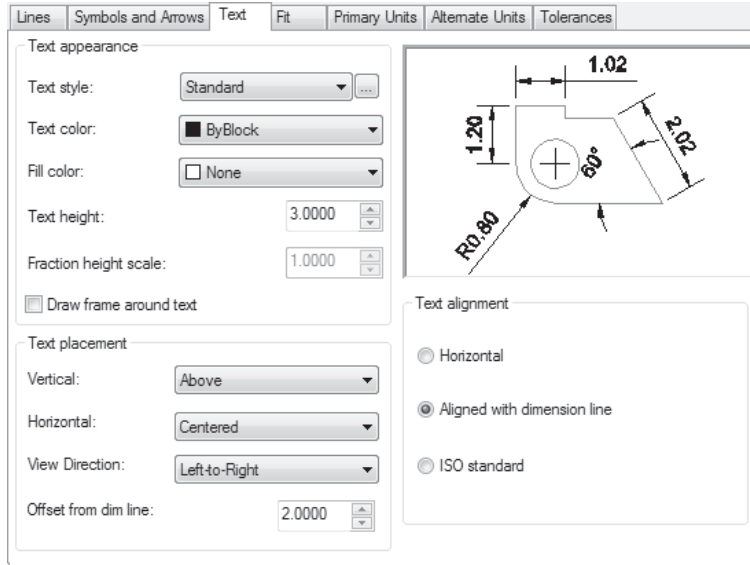


Linear Jog Dimension

Jog Height Factor specifies the size of the jog created by the DIMJOGLINE command.

TEXT

The **Text** tab specifies the format, placement, and alignment of dimension text. Additional settings affecting the display of dimension text are found in the Primary Units tab.



Text Appearance

Dimension text is treated independently of other text in the drawing. For example, dimension text uses the style defined by this dialog box, rather than the style defined by the **STYLE** command.

Text Style

The **Text Style** droplist selects a style previously defined by the **STYLE** command. The default is “Standard.” Choose the ... button to display the Text Style dialog box, which lets you create and modify text styles. The dimension text style name is stored in dimvar **DIMTXSTY**.

Text Color

The **Text Color** droplist selects the color of the dimension text (**DIMCLRT**). To choose a color not listed, pick **Select Color** at the bottom of the droplist.

Fill Color

The **Fill Color** option determines the color of a rectangle placed behind the dimension text (**DIMTFILL** and **DIMTFILLCLR**). One of the colors is called “Background,” which takes on the color defined by the **OPTIONS** command. (From the Tools menu, select **Options**, **Display**, and then **Colors**. The color for the model and layout backgrounds can be chosen separately.)



The **TEXTTOFRONT** command can make dimension text stand out from other objects.

Text Height

The **Text Height** option operates like the height setting in text styles: 0 means the same height as defined in the text style. Any other height entered here overrides the height defined by the style (**DIMTXT**).

Fraction Height Scale

The **Fraction Height Scale** option specifies the size of fractions and tolerances in dimension text, as a fraction of the dimension text (DIMTFAC). When the scale is 1, fractions and tolerances are the same height as the dimension text; when set to 0.5, fractions and tolerances are half the size of dimension text.

Draw Frame Around Text

The **Draw frame around text** check box adds a box around the dimension text (DIMGAP < 0). The distance between the box and the dimension text is specified later in the tab with the **Offset from dim line** option.

Text Placement

The **Text Placement** section specifies how dimension text is placed relative to the dimension line.

Vertical

The **Vertical** droplist specifies the vertical placement of dimension text (DIMITAD):

- **Centered** centers dimension text between extension lines and on the dimension line (default).
- **Above** places text above the dimension line.
- **Outside** places text on the side farthest from the extension line pick points.
- **JIS** places text according to the Japanese Industrial Standards for dimensions, above the dimension line.
- **Below** places text under the dimension line .

Horizontal

The **Horizontal** droplist specifies the horizontal placement of dimension text (DIMJUST).

- **Centered** centers text between the extension lines and on the dimension line (the default).
- **At Ext Line 1** left-justifies text against first extension line.
- **At Ext Line 2** right-justifies text against the second extension line.
- **Over Ext Line 1** positions text vertically over the first extension line.
- **Over Ext Line 2** positions text vertically over the second extension line.



View Direction

The **View Direction** droplist specifies the reading direction of dimension text (DIMTXTDIRECTION).

- **Left-to-Right** displays horizontal text left to right, and vertical text bottom to top.
- **Right-to-Left** displays horizontal text right to left, and vertical text top to bottom.

Offset From Dim Line

The **Offset from dim line** option specifies the gap between dimension text and the dimension line, as well as the space between the text and the frame (DIMGAP).

Text Alignment

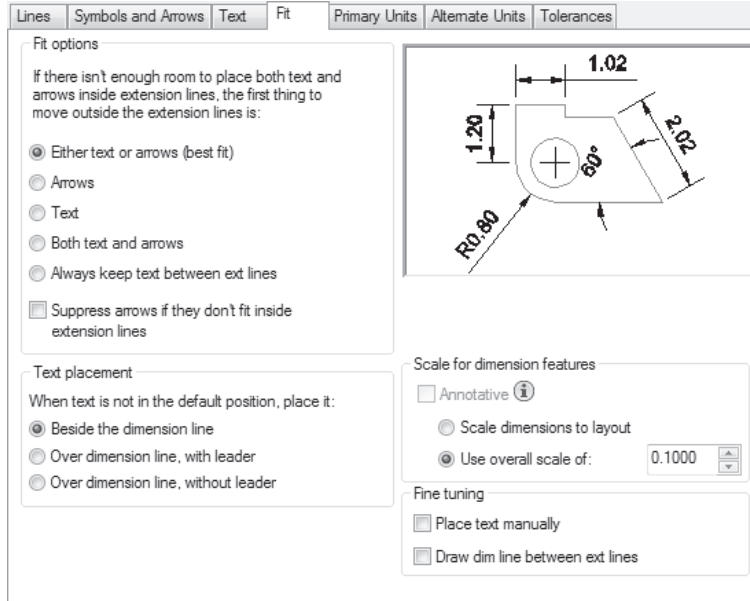
The **Text Alignment** option determines the orientation of dimension text when inside and outside the extension lines (DIMITH and DIMITOH).

- **Horizontal** forces text to be horizontal, whether or not it fits inside the extension lines.
- **Aligned with dimension line** forces text to align with the dimension line.

- **ISO Standard** aligns text with the dimension line when it fits inside extension lines; when outside extension lines, it is drawn horizontally.

FIT

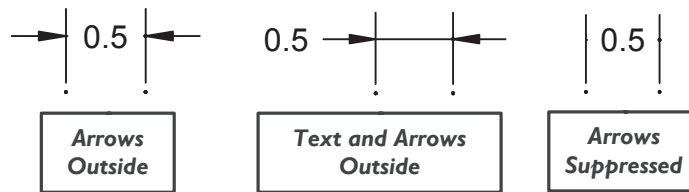
The **Fit** tab determines the placement of dimension text, arrowheads, leader lines, and dimension lines.



Fit Options

The **Fit Options** section controls where text and arrowheads are placed when the distance between extension lines is too narrow (**DIMATFIT**).

- **Either the text or the arrows, whichever fits best** places the elements where there is room.
- **Arrows** fits only the arrowheads between extension lines, when space is available.
- **Text** fits text between extension lines, but places arrowheads outside when space is lacking.
- **Both text and arrows** forces both text and arrows outside the extension lines.
- **Always keep text between ext lines** keeps text always between the extension lines (**DIMTIX**).



- **Suppress arrows if they don't fit inside the extension lines** draws no arrowheads in that case (**DIMSOXD**).

Text Placement

When AutoCAD cannot place the dimension text normally, you have these options (**DIMTMOVE**):

- **Beside the dimension line** places text beside the dimension line.
- **Over the dimension line, with a leader** draws a leader line between the dimension line and the text when there isn't enough room for the text.
- **Over the dimension line, without a leader** draws no leader.

Scale for Dimension Features

The **Scale for Dimension Features** section controls the size of text and arrowheads independently of the rest of the drawing, usually using the plot scale. The length of the dimension and extension lines is unaffected. (To change the weight of these lines, use the **Lineweight** option in the Lines and Arrows tab.)

Annotative toggles use of the annotative scaling property (**DIMANNO**). When on, the following two options become unavailable:

- **Use overall scale of** sets the scale factor for text and arrowheads (**DMSCALE**). Entering a value of 2, for example, doubles the size of arrowheads and text.
- **Scale dimensions to layout** scales dimensions to a factor based on the scale between model space and layouts, as follows:

AutoCAD sets **DIMSCALE** = 1 in model tab.

AutoCAD sets **DIMSCALE** = 0 in layout tab.

AutoCAD then computes **DIMSCALE** in model space viewports of layout tabs from the viewport's scale factor; or from the current value of the **ZOOM XP** command when dimensions are placed in paper space layouts.

Fine Tuning

The **Fine Tuning** section presents a couple of miscellaneous items:

Place text manually when dimensioning determines whether dimension commands prompt you for the position of the dimension text, allowing you to change the text position, if necessary (**DIMUPT**). AutoCAD ignores the horizontal justification settings.

Always draw dim line between ext lines forces AutoCAD always to draw the dimension lines inside the extension lines (**DIMTOFL**). The arrowheads and text are placed outside, if there isn't enough room. This option also forces leaders to be drawn to the circle's center point, even when the leader would not normally be drawn.

PRIMARY UNITS

The **Primary Units** tab sets the format of primary dimension units, as well as the prefix and suffix of dimension text. "Primary" refers to ordinary dimension text, as opposed to alternate units (shown in square brackets) and tolerance text.

The screenshot displays the **Primary Units** tab in the AutoCAD Dimension Style Manager. The **Linear dimensions** section includes:

- Unit format:** Decimal
- Precision:** 0.00
- Fraction format:** Diagonal
- Decimal separator:** . (Period)
- Round off:** 0.0000
- Prefix:** (empty)
- Suffix:** (empty)
- Measurement scale:** Scale factor: 1.0000, with an option to ☐ Apply to layout dimensions only.
- Zero suppression:**
 - ☐ Leading
 - ☐ Trailing
 - Sub-units factor:** 100.0000
 - ☒ 0 feet
 - ☐ 0 inches
 - Sub-unit suffix:** (empty)

 The **Angular dimensions** section includes:

- Units format:** Decimal Degrees
- Precision:** 0.00
- Zero suppression:**
 - ☐ Leading
 - ☒ Trailing

 A preview window on the right shows a mechanical part with dimensions: 1.02, 1.20, 2.02, and a 60-degree angle.

Linear Dimensions

The **Linear Dimensions** section specifies the dimensioning units, and whether the dimension text has a prefix or suffix. This section has features similar to those of the **UNITS** command. The Drawing Units dialog box controls the settings of units for all numbers, angles, and dimensions in the drawing; the Primary Units tab overrides those settings.

Units Format

The **Units format** droplist selects the format: scientific, decimal (default), architectural, engineering, fractional, and the units specified by the Windows operating system (**DIMLUIT**). The Windows units are changed by the Numbers tab of the Regional Options dialog box (found in the Control Panel).

Precision

The **Precision** droplist selects the number of decimal places or fractional divisions to be displayed. The range is from 0 to 8 decimal places, and from 0" to 1/256". The measured values are unaffected by this setting, which changes only the display.

Fraction Format

The **Fraction format** droplist specifies how fractions are stacked: horizontally, diagonally, or not at all (**DIMFRAC**). This option is available only when you select a units format with fractions, such as Architectural; otherwise, it is grayed out.

Decimal Separator

The **Decimal separator** droplist specifies the decimal separator: period (.), comma (,), or space () (**DIMDSEP**). This option is available only when you select **Decimal** as the units format. This is typically for countries that use the comma (,) to separate units from decimals.

Round Off

The **Round off** option specifies how to round off decimals and fractions of numbers; it does not apply to angles (**DIMRND**). For example, setting it to 0.33 rounds the dimension text to the nearest 0.33; the measured values are unaffected by this setting, which changes only the display. This option is different from **Precision**, which *truncates* the display of decimal places.

Prefix and Suffix

The **Prefix** and **Suffix** options provide room for any alphanumeric values, which are then added in front of and behind the dimension text. For example, to prefix every dimension with the word "Verify," enter that in the **Prefix** box. To suffix every dimension with "(TYPICAL)," enter that in the **Suffix** box. The **DIMPOST** dimvar stores both values using this format: "Verify<>(TYPICAL)".

Measurement Scale

The **Measurement Scale** section specifies the scale factor applied to the *value* of the dimension text; to change the size of text and arrows, specify the overall scale factor in the Fit tab.

Scale Factor

The **Scale Factor** option sets the scale factor that multiplies linear dimensions (**DIMLFAC**). By entering 25.4, for example, imperial dimensions are converted to millimeters (metric units). The factor is not applied to angles; alternate and tolerance values have their own scale factor settings.

Apply to Layout Dimensions Only

The **Apply to layout dimensions only** option applies the linear scale factor only to dimensions created in layouts (paper space). When turned on, the scale factor is stored as a negative value in **DIMLFAC**.

Zero Suppression

The **Zero Suppression** section determines whether zeros are displayed in dimension text.

Leading and Trailing

The **Leading** and **Trailing** options specify whether leading or trailing zeros, and the zero feet or inches, are suppressed (DIMZIN). When both options are turned on, **0.2500** becomes **.25**, and **1.00** becomes **1**. Note that zero suppression overrides the setting of the **Precision** option.



Sub-units

The following options operate only when zero-suppression is turned on, and with certain formats of units:

The **Sub-units Factor** option specifies the conversion factor, such as 100 for cm-to-m, or 12 for in-to-ft.

The **Sub-unit Suffix** option specifies the subunit, such as centimeters for meters, or inches for feet.

0 Feet and 0 Inches

The **0 feet** and **0 inches** options determine whether zero feet and zero inches are displayed. When both options are turned on, **0'-1/2"** becomes **1/2"** and **34'-0"** becomes **34'**. These feet and inches options become available only when a fraction unit format is selected, such as Architectural.

Angular Dimensions

The Angular Dimension section specifies the display format for dimensions placed by the DIMANGULAR command. This allows angles to be formatted independently of linear dimensions.

Units Format

The **Units format** droplist specifies the format of angular dimensions (DIMAUNIT) — curiously, the Surveyor's units are missing:

- Decimal degrees (*DdD.dddd*, the default).
- Degrees/ Minutes/Seconds (*DD.MMSSdd*).
- Grads (*DDg*).
- Radians (*DDr*).

Precision

The **Precision** droplist specifies the number of decimal places or fractions (DIMADEC). This affects the display of the angle and not its measured value.

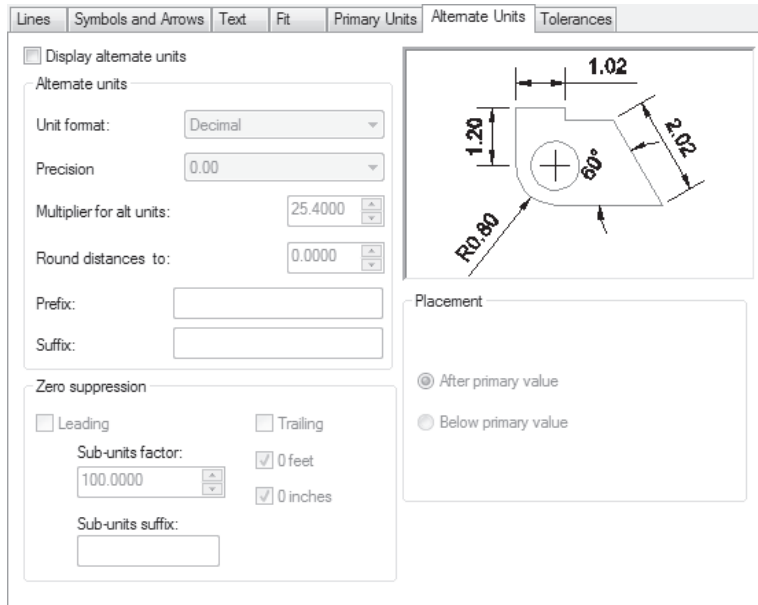
Zero Suppression

The **Zero Suppression** section suppresses leading and trailing zeros of angles (DIMAZIN). As an example of suppressing leading zeros, the measured angle of **56°07"** is displayed as **56°7"**; and in suppressing trailing zeros, **56°00"** is displayed as **56°**.

ALTERNATE UNITS

The **Alternate Units** tab selects the format of units, angles, dimensions, and scale of alternate measurement units. The options are similar to the Primary Units tab; the following descriptions list only the differences.

AutoCAD allows you to place dimensions with double units: a primary unit plus a second or *alternate* unit. The alternate units appear in square brackets. This is particularly useful for drawings that must show imperial and metric units. Angles cannot have alternate units in AutoCAD.



When you select options in this tab, the values are stored in a different set of dimvars:

DIMALTD specifies the decimal places for alternate units.

DIMALTRND specifies the rounding of alternate units.

DIMALTU specifies the format of alternate units, except for angular dimensions.

DIMALTZ specifies zero suppression for alternate units.

The **Display alternate units** check box turns on alternate units (DIMALT); when off, all options are grayed out, meaning they are unavailable.

Alternate Units

The **Alternate Units** section specifies the look of the alternate text; the options are identical to that of the Primary Units tab.

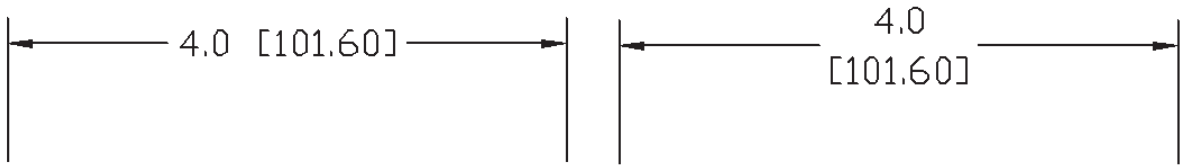
Zero Suppression

The **Zero Suppression** section determines whether leading and trailing zeros are suppressed; the options are identical to that of the Primary Units tab.

Placement

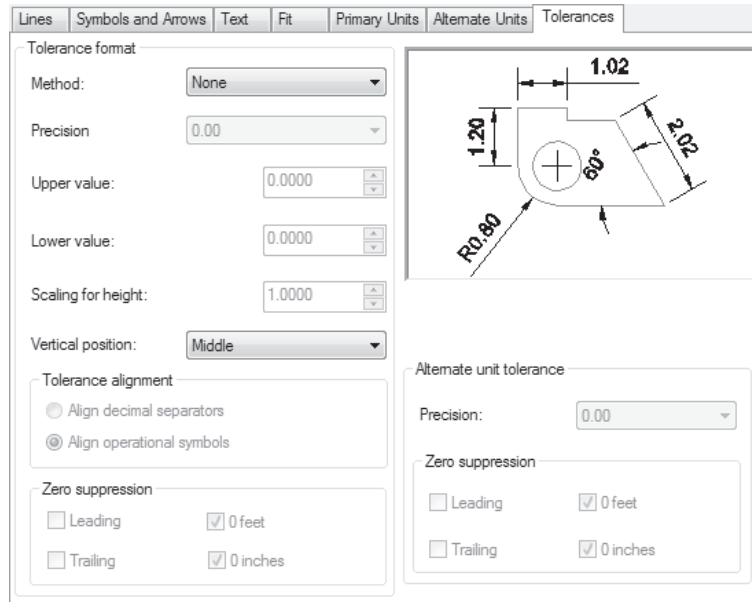
The **Placement** section determines where the alternate units are placed (DIMAPOST):

- **After primary value** places the alternate units behind the primary units.
- **Below primary value** places the alternate units below the primary units



TOLERANCES

The **Tolerances** tab controls the format of dimension text tolerances. It does not format tolerance symbols placed with the TOLERANCE command; they have no format options.



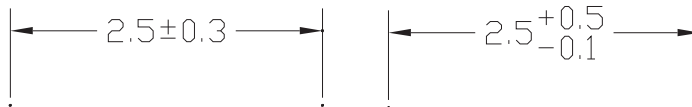
Tolerance Format

The **Tolerance Format** section determines the style and look of tolerances.

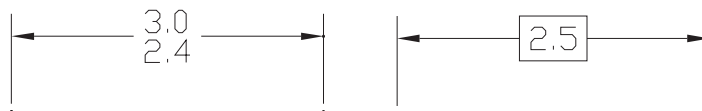
Method

The **Method** droplist offers five styles of tolerance text (**DIMTOL** and **DIMLIM**):

- **None** suppresses the display of tolerance text (default), and grays out all options in this tab, except **Vertical position**, curiously enough.
- **Symmetrical** adds a single plus/minus tolerance, such as the 0.3 illustrated below.



- **Deviation** adds a plus and a minus tolerance, such as the +0.5 and -0.1 illustrated above.
- **Limits** places two dimensions, such as the 3.0 and 2.4 illustrated below. AutoCAD arrived at these values by adding 0.5 to 2.5 (=3.0), and subtracting 0.1 from 2.5 (=2.4).



- **Basic** draws a box around the dimension text. (The distance between text and box is stored in **DIMGAP**.) No tolerances are displayed, as illustrated above.

Dimvar **DIMTOL** appends the tolerances to the dimension text; **DIMLIM** displays dimension text as limits. Turning on **DIMLIM** turns off **DIMTOL**.

Precision

The **Precision** droplist determines the number of decimal places displayed by both tolerance values.

Upper Value and Lower Value

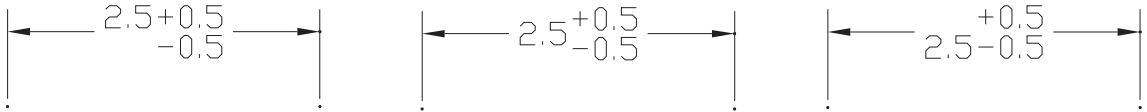
The **Upper Value** and **Lower Value** options specify the upper or plus value (DIMTP), and the lower or minus tolerance value (DIMTM).

Scaling for Height

The **Scaling for height** option determines the size of the tolerance text relative to the main dimension text (DIMTFAC). A value of 1 indicates each character of tolerance (or limits) text is the same size as the primary dimension text. Changing the value to **0.5** reduces the tolerance text height by half.

Vertical Position

The **Vertical position** droplist determines the relative placement of tolerance text (DIMITLJ), whether it is aligned to the top, middle, or bottom of the primary dimension text.



Tolerance Alignment

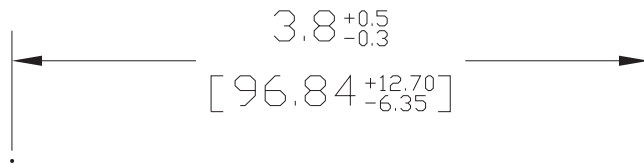
The **Tolerance Alignment** options align the deviation and limits tolerance text by decimal points or prefix symbols, such as + and -.

Zero Suppression

The **Zero Suppression** options are identical to those found in the Primary Units tab.

Alternate Unit Tolerance

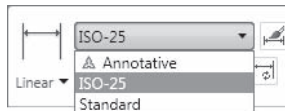
The **Alternate Unit Tolerance** option specifies the number of decimal places for the tolerance values in the alternate units of a dimension (DIMALTTD). This option is available only when alternate units are turned on in the Alternate Units tab. Combining tolerances with alternate units leads to cluttered dimension text.



SETTING THE CURRENT DIMSTYLE

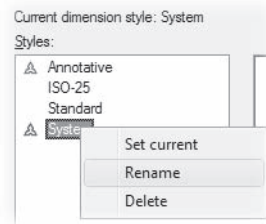
To set a dimension style as active, select the dimstyle name from the list under Styles, and then choose the **Set Current** button. (The current dimension style name is stored in dimvar DIMSTYLE.)

Alternatively, you can use the ribbon's Dimensions panel or the Styles toolbar, which provide droplists of dimension and text styles in the drawing. To make a dimstyle current, select it from the droplist.



Renaming and Deleting Dimstyles

It is not immediately apparent how to rename or delete dimension styles in the Dimension Style Manager. The commands are “hidden” in shortcut menus: select a dimension style name in the **Styles** list. Right-click to display the cursor menu, and then choose **Rename** or **Delete**.



Not every style can be erased or renamed. If a style is in use, it cannot be erased. The “Standard” dimstyle can be renamed, but not deleted. Externally-referenced dimstyles cannot be renamed or deleted.

Alternatively, you can use the **RENAME** command to rename dimstyles, and the **PURGE** command to remove unused ones.

MODIFYING AND OVERRIDING DIMSTYLES

After you create one or more dimension styles for drawings, you might need to change the style. Perhaps the client wants dimension lines thicker than the extension lines, or the units changed to metric, or perhaps some extension lines removed.

Some changes are *global*, which means they apply to every dimension in the drawing, such as the thicker dimension lines. Other changes are *local*, which means they apply to selected dimensions, such as extension lines to be removed.

Some changes are *retroactive*, which means they apply to all dimensions already drawn. Other changes are *proactive*, which means they apply to dimensions not yet drawn.

Here is how AutoCAD handles these four cases.

To apply global changes retroactively: you modify the dimension style with the **DIMSTYLE** command's **Modify** button. AutoCAD retroactively changes all dimensions in the drawing.

To apply global changes proactively: you modify the dimension style with the **DIMSTYLE** command's **Override** button, which creates “sub-styles.” The next dimension you draw reflects the change in dimension style.

To apply different dimension styles to selected dimensions (local changes): you use the **-DIMSTYLE** command's **Apply** option to select dimensions, and then apply the dimstyle to them — a local modification.

To change dimension variable settings of selected dimensions (local changes): you use the **DIMOVERRIDE** command to specify the name of a dimension variable and then provide the new value. It prompts you to select the dimensions to which the local override should be applied.



Note The difference between *modifying* and *overriding* dimension styles:

Modify — modifies retroactively, and globally updates all existing dimensions created using that style.

Override — changes only dimensions created after the override is applied,

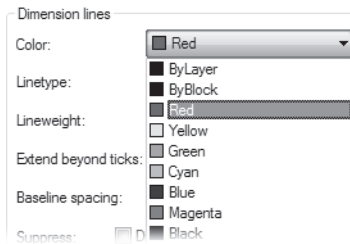
TUTORIAL: GLOBAL RETROACTIVE DIMSTYLE CHANGES

In the following tutorial, you learn how to change the style of all dimensions in the drawing at once.

1. Start a new drawing with AutoCAD, and then draw some dimensions.
2. To change the style of all dimensions in the drawing, start the **DIMSTYLE** command:
3. In the dialog box, click **Modify**.



4. In the Lines tab, change the dimension line Color to red.



Notice that the preview shows the dimension lines changed to red.

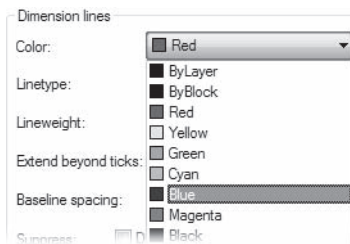
5. Click **OK**, and then **Close**.

Notice that all dimensions in the drawing now have red dimension lines.

TUTORIAL: GLOBAL TEMPORARY DIMSTYLE CHANGES

In this, you override the dimension style temporarily, so that new dimensions take on the changed style; existing dimensions remain unaffected.

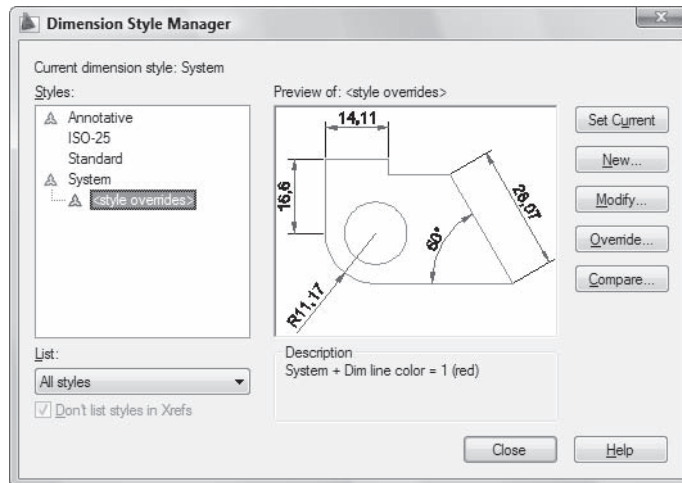
1. Continue with the drawing from the previous tutorial.
2. Start the **DIMSTYLE** command, and then choose the **Override** button.
3. In the Lines tab, change the dimension line color to blue.



4. Click **OK**.

Notice that the Standard dimstyle has a sub-style called "<style override>."

It is highlighted, meaning that it is current. (After exiting this dialog box, any dimension you draw has blue dimension lines.)



5. In the Description area, notice the change to the blue dimension lines: “Standard+Dim line color = 5 (blue)”.
6. Click **Close**. Notice that none of the existing dimensions in the drawing changes.
7. Draw a linear dimension with the **DIMLINEAR** command. Notice that its dimension line is blue.

Unfortunately, the names of overrides are not listed by the droplist in the ribbon’s Annotation panel or Styles toolbar.



Note You can turn overrides into permanent dimension styles. Here’s how:

To turn overrides into new dimstyles, right-click “<style override>” (the AutoCAD-generated name), and then select **Rename** from the shortcut menu. Change the name to something meaningful, like “Blue Dimlines.”

To make an override the current dimstyle, right-click “<style override>”, and then select **Save to current style** from the shortcut menu. Notice that the override disappears, and that the current style shows the changes, such as blue dimension lines. Click **Close**, and all dimensions change to blue dimlines. (This is equivalent to using the **Modify** button in the first place.)

STYLE OVERRIDES, DIMOVERRIDE, AND PROPERTIES

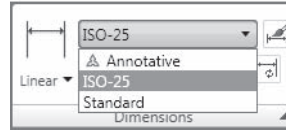
Overriding individual dimensions is called “local” changes.

The Dim Style droplist on the Dashboard’s Dimension panel applies dimension styles to selected dimensions, as does the droplist on the Styles toolbar. The **DIMOVERRIDE** command applies changed dimension variables to selected dimensions.

TUTORIAL: LOCAL DIMSTYLE CHANGES

1. For this tutorial, create a drawing with at least two dimension styles and several dimensions.
2. To apply a dimstyle to certain dimensions, select a few dimensions in the drawing. Notice that they are highlighted.

- From the ribbon's Dimensions panel, click the **Dimension Style** droplist. (You can also do the same with the Dimension Style droplist in the Styles and Dimension toolbars.) Notice the list of dimension style names.



- Select a different style name. Notice that the selected dimensions change their style.
- Press **ESC** to remove the highlighting from dimensions.

TUTORIAL: LOCAL DIMVAR OVERRIDES

In this tutorial, you change the color of one dimension's extension line to yellow.

- For this tutorial, continue with the drawing from the last tutorial, the one with at least two dimensions styles.
- To change a dimvar for selected dimensions, start the **DIMOVERRIDE** command.
 - At the 'Command:' prompt, enter the **dimoverride** command.
Command: **dimoverride** (Press ENTER.)
 - Alternatively, enter the aliases **dov** or **dimover** at the 'Command:' prompt.
- Enter the name of a dimvar, such as **DIMCLRE**, which sets the color of extension lines:
Enter dimension variable name to override or [Clear overrides]: **dimclre**
- Enter a new value for the dimvar, such as yellow:
Enter new value for dimension variable <BYBLOCK>: **yellow**
- The prompt repeats so that you can override other dimvars. Press **ENTER** to continue and select the dimensions to override:
Enter dimension variable name to override: (Press ENTER to continue.)
- Select the dimensions to change:
Select objects: (Select a dimension.)
Select objects: (Press ENTER to end object selection.)
Notice that the extension lines change to yellow.



Note You can override dimvars *during* dimensioning commands. In the following example of the **DIMALIGNED** command, the dimension line color is changed to green using the **DIMCLRD** dimvar:

Command: **dimaligned**

Specify first extension line origin or <select object>: **dimclrd**

Enter new value for dimension variable <byblock>: **green**

Specify first extension line origin <select object>: (Continue with the **DIMALIGNED** command.)

New dimension lines are now green, until you override again.

Clearing Local Overrides

If you change your mind, you can clear local overrides. Start the **DIMOVERRIDE** command, and then enter the **Clear overrides** option:

Command: **dimoverride**

Enter dimension variable name to override or [Clear overrides]: **c**

Select objects: *(Select the dimensions to return to normal.)*

Select objects: *(Press ENTER to end object selection.)*

The dimension returns to its previous look, because the override is canceled.

PROPERTIES

You can use the Properties palette to change many properties of dimensions locally. Open the palette with the **PROPERTIES** command, and then select the dimensions you wish to change.

DRAWORDER FOR DIMENSIONS

You can force AutoCAD to display dimensions in front of all other objects in the drawing. To do this, use the **TEXTTOFRONT** command, as follows:

Command: **texttfront**

Bring to front [Text/Dimensions/Both] <Both>: **d**

n object(s) brought to front.

This command is more efficient than the **DRAWORDER** command, because it selects all dimensions for you.

Because **TEXTTOFRONT** is a command, and not a setting, you may have to reuse this command as the drawing progresses.

GRIPS EDITING

Grips allow you to edit dimensions and leaders directly — without needing to enter the names and options of dimension editing commands.

The grips on associative dimensions change the location of the dimension line, text, extension lines, and so on. The tricky part is knowing which grip performs what action; the grips themselves provide no clues, unfortunately. Thus, this section describes their actions.

TUTORIAL: EDITING DIMENSIONS THROUGH OBJECTS

1. Draw a line, and then dimension it using the **Select Object** option, as follows:

Command: **dimlinear**

Specify first extension line origin or <select object>: *(Press ENTER.)*

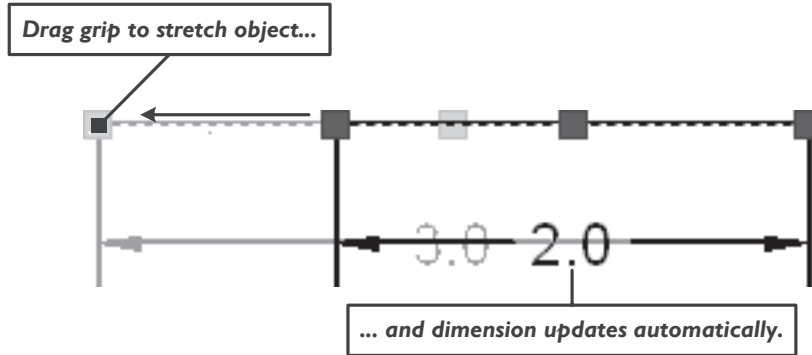
Select object to dimension: *(Pick the line.)*

Specify dimension line location or [Mtext...]: *(Pick a point to position the dimension line.)*

Dimension text = 2.00

2. Select the line — not the dimension! Notice the blue grips.
3. Select one of the end grips; it turns red.

4. Stretch the *object*, longer or shorter. Notice that the associated dimension updates itself, as illustrated by the figure.



5. Select middle grip to move the line. Notice that the dimension tags along.
6. Press **ESC** to end grips editing.

GRIPS EDITING OPTIONS

Grips perform six editing operations: stretch, copy, move, rotate, scale, and mirror. Here is how each one affects dimensions:

Stretch changes the position of dimension parts, such as extension lines or text, while keeping the remainder of the dimension in place. The dimension remains partially associated with its object. When stretching the dimension line, the dimension text is updated; other parts of the dimension react differently to stretching, as detailed later.

Copy copies the dimension, subject to the peculiarities of the selected grip.

Move moves the entire dimension, but disassociates it from its object.

Rotate rotates the entire dimension about the selected grip, but disassociates it from its object.

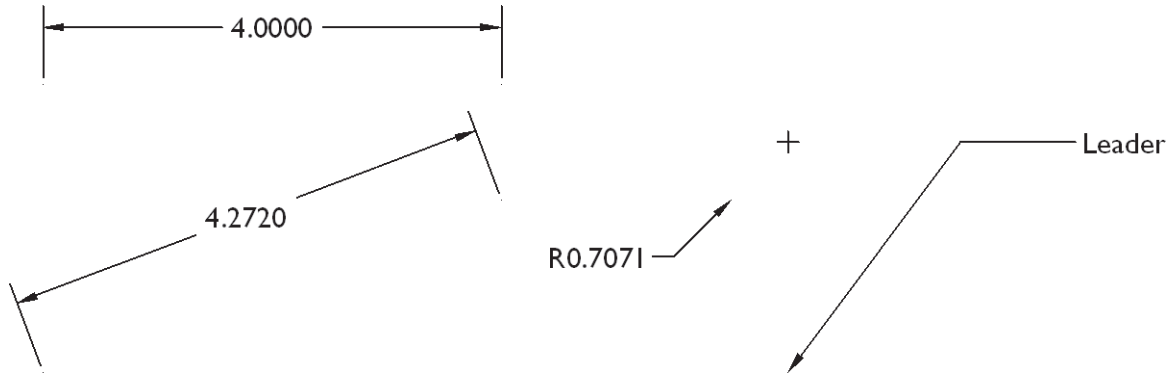
Scale resizes the dimension, using the selected grip as the base point. The dimension remains associated with its object; dimension text is updated to reflect the new size. As with copying, scaling is subject to the peculiarities of the selected grip.

Mirror does not mirror dimensions; instead, this option rotates them about the selected grip. Text is not mirrored, ignoring the setting of the **MIRRTEXT** system variable. The “mirrored” dimension is disassociated from its object.

EXERCISES

- I. In the following exercises, you modify the Standard dimension style.
 - a. Start AutoCAD with a new drawing.
 - b. Draw one each of the following dimensions — linear, aligned, diameter, and leader — similar to the figure below.

Which commands did you need to draw each dimension?



- c. Open the Dimension Style Manager dialog box. Which dimstyles are listed there? Modify the dimstyle: change the color of extension lines to red. When you exit the dialog box, what happens?
 - d. Return to the Dimension Style Manager dialog box. Override the dimstyle: change the color of text to blue. When you exit the dialog box, what happens?
 - e. Draw another linear dimension. Is it different from the others?
 - f. Return to the Dimension Style Manager dialog box. Convert the override into a dimstyle called “Blue Text.” When you exit the dialog box, what happens?
 - g. Use the Ribbons’s Annotation tab to apply the Blue Text dimstyle to the aligned dimension. Does it change? Repeat, applying the dimstyle to the leader. Does it change?
2. Continuing with the drawing from above, create a new dimension style with the following settings:

Extension line:

Color 8

Text:

Font Times New Roman

Color 155

Vertical Placement JIS

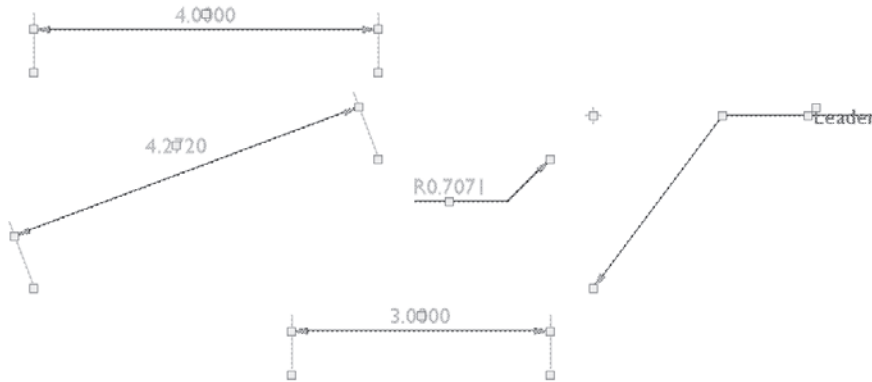
Arrowheads:

Both Architectural Tick

Hint: change the text style.

Save the style by the name of “JayGray.” Apply the style to all dimensions in the drawing. Save the drawing as *jaygray2.dwg*.

3. Continuing with the drawing from the previous exercise, use grips editing on the different grips of each dimension. Do not save the drawing.



4. Start a new drawing.
Open the DesignCenter, and then import dimension styles from the following drawing:
Architectural - Annotation Scaling and Multileaders.dwg.
5. From AutoCAD's \sample folder, open the *Architectural - Annotation Scaling and Multileaders.dwg* file, a drawing of a set of stairs.
Select a dimension, and then edit its text
Do the same to a leader.
6. Start a new drawing, and then draw a linear dimension.
 - a. Add a jog to the dimension line.
Which command did you use to add the jog?
 - b. Draw a second linear dimension that crosses the first one.
Add a gap where the two dimensions cross.
Which command did you use to create the gap?
7. From the Companion CD, open *structure.dwg* file, the drawing of an arbour. Space the linear dimensions evenly.
Which command did you use to space the dimensions?

CHAPTER REVIEW

1. Which command is used to create, edit, and delete dimension styles?
2. What is the difference between *modifying* and *overriding* dimensions?
3. Can the color and lineweight of dimension lines be set independently of extension lines?
4. Can extension lines be suppressed independently of each other?
5. Is it better to use dimension styles or layers to set the color and lineweight of dimension lines?
6. Can you define your own arrowhead for use by dimensions?
7. When the distance between extension lines is too tight, where does AutoCAD place the dimension text?
8. Describe some situations where you might need to edit dimensions.
9. What are *alternative units*?
10. When you move an object, do associative dimensions move also?
11. Describe how the following grips editing commands affect dimensions:
 - Stretch
 - Copy
 - Move
 - Rotate
 - Scale
 - Mirror
12. What is the purpose of the **AIDIMFLIPARROW** command?
13. Can extension lines have fixed lengths?
14. Explain the meaning of the following abbreviations:
 - dimvar
 - dimstyle
 - dimline
15. What is the “T” in **DIMTEDIT** short for?
16. Describe the function of the **DIMEDIT** command’s **Oblique** option. When is it useful?
17. How do you edit the wording of the dimension text?
 - The properties of dimension text?
 - The position of dimension text?
18. Briefly explain the differences among these three similar-sounding dimension-editing commands:
 - DDEDIT**
 - DIMEDIT**
 - DDIMTEDIT**
19. If you make a mistake editing a dimension, which command returns the original dimension?
20. Can dimensions be scaled independently of the drawing?
21. What is the danger in rounding dimension text?
22. When AutoCAD rounds off dimension values, is the change permanent?

23. Round the following numbers to the nearest unit:
 2.5
 1.01
 4.923
 10.00
 6.489
24. Where does the JIS dimension standard expect text to be placed?
25. Describe the purpose of *dimension variables*.
26. Does every drawing have a dimension style?
27. How do you change the scale of dimensions relative to the rest of the drawing?
28. List two sources of dimension styles:
 - a.
 - b.
29. Why might you want to edit dimensions in the following manner:
 - a. Create gaps in extension lines?
 - b. Create jogs in the dimension line?
 - c. Space dimensions evenly?
30. Briefly describe the purpose of the following dimvars. (For help, look up the tables in appendix D.)
 - DIMCLRE**
 - DIMSEI**
 - DIMJUST**
 - DIMCLRT**
 - DIMAFIT**
31. Find the name of the dimvar that does the following task. (For help, look up the tables in appendix D.)
 - Specifies the size of the center mark.
 - Enables alternate units.
 - Specifies the length of the arrowhead.
 - Specifies linear dimension units.
 - Controls suppression of zeros.
32. Why does the Standard dimension style set **DIMALTF** (alternate scale factor) to 25.4?
 And why does the ISO-25 dimstyle set the same dimvar to 0.04?
33. What is the effect of annotative scaling on dimensions?
34. What is the command that corresponds to the alias?
 - ddim
 - dimted
 - ded
 - dov
 - d
35. Can you rename dimension styles?
 Purge dimension styles?
36. Can a drawing have more than one dimension style?
 Can a dimension have two different arrowheads?
37. Can you create your own arrowheads for dimensions?

38. Under what condition do the arrowheads and text appear outside the extension lines?
39. What is the purpose of *zero suppression*?
Why might you want to use zero suppression?
40. When are alternative units used?
How are alternative units shown?
41. Briefly describe the sequence to change the style of all dimensions in the drawing retroactively.
42. Briefly describe the sequence to change the style of dimensions selectively.
43. Can dimvars be changed during a dimensioning command?
44. If dimensions are hidden by other objects in the drawing, what can you do to see them better?
45. How would you quickly bring all dimensions to the top of the display order?

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CHAPTER 13

Applying Dimensional Constraints

AutoCAD's dimensions report the sizes of objects, as you saw in the previous chapters. A special type of dimension, called the dimensional constraint, controls the size of objects and the distances between them. In this chapter, you learn about dimensional constraints; in the following chapter, you learn how to control the geometry in and between objects using geometric constraints.

Typically, you apply geometric constraints to objects first, and then apply dimensional constraints. But I find constraints are best approached by learning about dimensional ones first.

In this chapter, you learn the following commands:

DIMCONSTRAINT applies dimensional constraints within and between objects.

DELCONSTRAINT erases dimensional constraints.

PARAMETERS edits links between dimensional constraints.

CONSTRAINTSETTINGS specifies default conditions for dimensional constraints.

NEW TO AUTOCAD 2010 IN THIS CHAPTER

- All of the commands listed above are new to AutoCAD 2010.



DIMCONSTRAINT

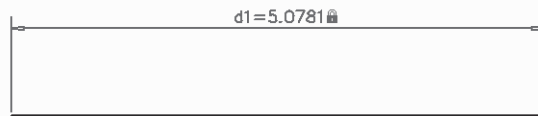
The **DIMCONSTRAINT** command applies dimensional constraints between objects and to geometric features within objects.

This command operates almost exactly like the dimensioning commands you learned in the earlier chapters. The difference is that this single command handles all forms of dimensional constraints, instead of one command handling each type of dimension like **DIMLINEAR** and **DIMRADIAL**.

Through options, this command supports these types of dimensional constraints: linear, horizontal, vertical, aligned, angular, radial, and diameter. In addition, it converts regular dimensions into dimensional constraints, and changes the form of dimensional constraints between dynamic and annotational.

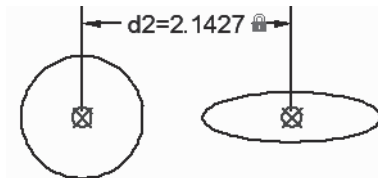
ABOUT DIMENSIONAL CONSTRAINTS

Dimensional constraints define the sizes of objects, such as the lengths of lines and radii of circles. For example, when you change the value of the linear dimensional constraint in the figure below, the line is forced to change its length. You cannot do this with regular dimensions, for they only update their values when the associated geometry is changed.



Next to the text is a padlock icon. It indicates that the dimension is a constrained one. If you find the icon annoying, it can be turned off.

Dimensional constraints can also be applied between points on two objects. For example, the distance between the center of the ellipse and circle is dimensionally constrained.



Dimensional constraints contain values (numbers) like regular dimensions; what's better, they can contain formulas and references to other dimensional constraints. (In the figures above, the 'd1=' and 'd2=' are simple formulas.) The Parameters Manager palette edits formulas and defines relationships.

Dimensional constraints can be used with *geometric constraints* to create *parametric drawings*. These drawings change their design and shape depending on the values you enter. (Geometric constraints are discussed in the following chapter.) AutoCAD is limited to 2D constraints, although they can be used in 3D to a limited extent, such as to control elevation and thickness.

(Historical note: Parametric CAD was invented by two Russian computer programmers in the mid-1980s. In the USA, they formed a software company named Parametric Technology Corp to market and sell the software, called Pro/Engineer, to mechanical engineers. Some 15 years later, they left PTC to develop Revit, parametric software for architects. Revit is now owned by Autodesk. AutoCAD also copies the constraint naming convention used by most parametric modelers, such as d1 and rad1.)

Dimensional constraints cannot be converted back to regular dimensions.

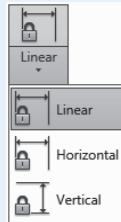
USING THE RIBBON WITH CONSTRAINTS

Autodesk has designed the **DIMCONSTRAINTS** command in such a way that it is easier to use with the ribbon than at the command prompt. Dimensional constraints are found in the Parametric tab's Dimensional panel:



DIMENSIONAL PANEL

 **Linear** applies linear dimensional constraints; they can be horizontal or vertical, depending on how you drag the extension lines — similar to the DimLinear command. This button is a flyout that holds the following options:



 **Horizontal** applies horizontal dimensional constraints.

 **Vertical** applies vertical dimensional constraints.

 **Aligned** applies aligned dimensional constraints at an angle.

 **Radial** applies radial dimensional constraints to arcs, circles, and polylines, like the DimRadial command.

 **Diameter** applies diameter dimensional constraints to arcs, circles, and polylines, like the DimDiameter command.

 **Angular** applies angular dimensional constraints between two lines, on arcs, and on vertices, like the DimAngular command.

 **Form** toggles dimensional constraints between dynamic and annotational.

 **Show Dynamic Constraints** toggles the display of dimensional constraints.

Click the  arrow to access the Constraint Settings dialog box; see the **CONSTRAINTSETTINGS** command.

MANAGE PANEL

 **Delete Constraints** removes constraints from all or selected objects; see the **DELCONSTRAINTS** command.

 **Parameters Manager** displays the Parameters Manager palette for creating and editing dimensional constraint formulas; see the **PARAMETERS** command.

(The buttons in the Geometric panel relate to geometric constraints.)

Types of Dimensional Constraints

AutoCAD has four types of dimensional constraints:

Dynamic constraints display in model tab only; they do not display in layouts, and they are not plotted. They are displayed in a unique and uneditable dimension style to ensure that they always appear the same size, no matter the zoom level. This is the default style of dimensional constraint, and the type you use the most in this chapter.

Annotational constraints act just like regular dimensions. They are displayed in model tab and in layouts. They are plotted and are affected by dimension styles. Their size changes with zoom levels.

To create annotational dimensions from scratch, set system variable `CCONSTRAINTFORM=1`, or change the `DIMCONSTRAINT` command's **Form** option to "Annotational." You can switch dimensional constraints between dynamic and annotational with the **PROPERTIES** palette's **Constraint Form** property.

Reference constraints refer to constraints that do not constrain geometry on their own. They are useful for reporting the overall widths of models, the value being the sum of two or more dynamic constraints. As the values of dynamic constraints change, the values reported by references are updated; you cannot edit the value of reference constraints.

Reference constraints are created automatically when a drawing is *overconstrained* — when there are too many dimensional constraints applied to the same set of objects. Reference constraints are identified by the parentheses that surround the text.

Parametric constraints are like dynamic constraints, but also sport user-definable grips for controlling the size of the related dynamic blocks. These are created in the Block Editor with the `BCPARAMETERS` command.

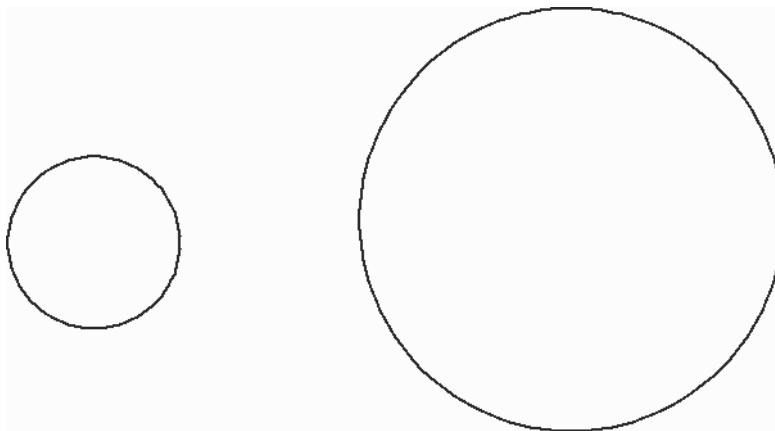


Note If you intend to plot drawings with dimensional constraints, you should primarily use them in annotational form.

TUTORIAL: PLACING DIMENSIONAL CONSTRAINTS

The best way to understand dimensional constraints is to try them out. In this tutorial, you place a dimensional constraint on one circle and then make its radius equal to that of a second circle.

1. Start AutoCAD with a new drawing, and then draw two circles with the **CIRCLE** command. Their sizes do not matter, except that one should be one larger than the other

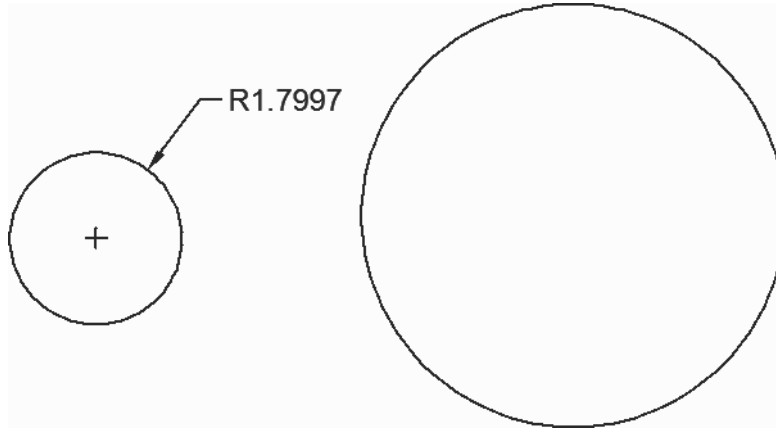


2. Dimension one of the circles with the **DIMRADIUS** command. (In the next step, you convert this dimension into a dimensional constraint.)

Command: **dimradius**

Select arc or circle: *(Pick one arc, circle, or polyarc.)*

Specify dimension line location or [Mtext/Text/Angle]: *(Pick a point to position the dimension leader.)*



3. To convert regular dimensions to dimensional constraints, start the **DIMCONSTRAINT** command, as follows:

- Enter the **dimconstraint** command, as follows:

Command: **dimconstraint**

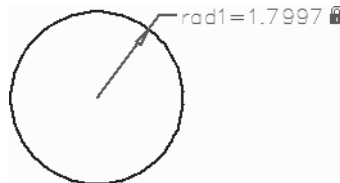
- Alternatively, enter the **DCON** alias.

4. At the prompt, pick the radial dimension that you drew earlier:

Current settings: Constraint form = Dynamic

Select a dimension or [LInear/Horizontal/Vertical/ALigned/ANGular/Radial/Diameter/Form <Aligned>]: *(Pick the radial dimension.)*

Notice that the dimension text changes to **rad1=1.7997**, and that the padlock icon is added. (The number you see may differ.) The term “rad1” is short for “radial,” and the 1 means that this is the first radial dimensional constraint in the drawing.



The other item to note is that the dimension style changes to one that is unique to dimensional constraints. The color is gray and the scale is annotative, which means the arrowhead and text display the same size, no matter the zoom level of the drawing; you cannot edit this dimension style.

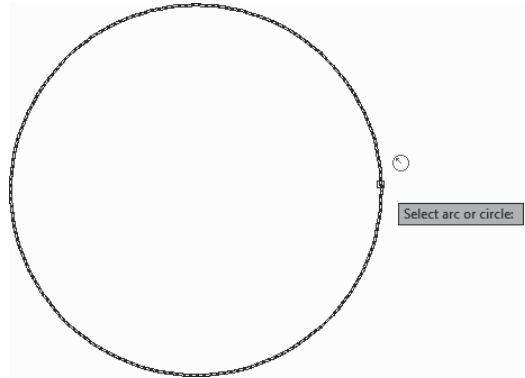
5. Place a constrained radial dimension on the second circle. To do this, repeat the **DIMCONSTRAINT** command but this time enter **R** as the option:

Command: *(Press spacebar.)*

DIMCONSTRAINT Current settings: Constraint form = Dynamic

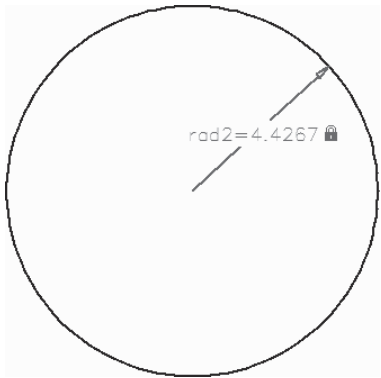
Select a dimension or [LInear/Horizontal/Vertical/ALigned/ANGular/Radial/Diameter/Form <Aligned>]: *(Type R.)*

6. Follow the prompts to place the dimensional constraint:
Select arc or circle: *(Pick the second circle.)*



To remind you that you are placing a radial constraint, AutoCAD displays an icon near the cursor.

Specify dimension line location: *(Pick a point to position the dimension leader.)*
*(Press **Enter** to end the command.)*



Notice that the dimension text reads “rad2,” meaning this is the second radial constraint.



Note AutoCAD automatically assigns names according to the type of dimensional constraint; you may edit them with different names. AutoCAD’s predefined names are as follows:

Name	Meaning
d	Distance (length)
rad	Radius
dia	Diameter
ang	Angle
user	User-defined

7. Try changing the radius of one of the circles:
- a. Select the second circle (rad2).
 - b. Select a grip, and then attempt to drag it to make the circle larger.

Notice that the circle won’t change. That’s because the dimensional constraint locks the radius, preventing you from changing the circle’s size — as indicated by the padlock icon.



Note To “unlock” the circle’s radius temporarily, follow these steps:

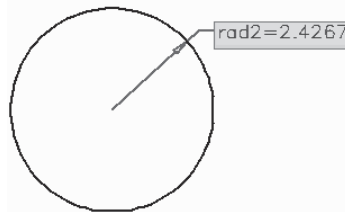
- a. Select the circle, and then click the center grip; it turns red.
- b. Tap the **Ctrl** key.
- c. Drag the grip; the circle changes size, and the dimensional constraint updates its value

(When you drag a circle’s quadrant grip, AutoCAD places an identical copy on top of the first.)

AutoCAD reacts differently when you *hold down* the **Ctrl** key while dragging the grip: a copy of the circle is made, complete with constraint(s). The constraint identifier is updated, such as rad3 or rad4.

The **Ctrl** key operates similarly with all other types of dimensional constraints.

8. To change the circle’s radius, you edit the value of the dimensional constraint:
 - a. Double-click the text of the dimensional constraint. Notice that you can edit it.
 - b. Change the value of the radius, leaving alone the “rad2=” prefix.



- c. Press **Esc** to exit editing mode. Notice that the circle changes its radius.
(You can also use the **TEXTEDIT** command to edit dimensional constraint text.)



Note The text displayed by dimensional constraints comes in two parts: a *name* and an *expression*.

- *Name* identifies the constraint dimension (such as rad2).
- *Expression* is a value that reports the distance (such as =2.4267).
- *Parameter* is the name (rad2) and expression (=2.4267) together. It can also be a formula that calculates the distance, such as dia1 = 2 x dia2 — one circle’s diameter is 2 times the diameter of another.

The ConstraintSettings dialog box determines whether the drawing shows names, expressions, or both.

Leave the drawing open for the next tutorial.

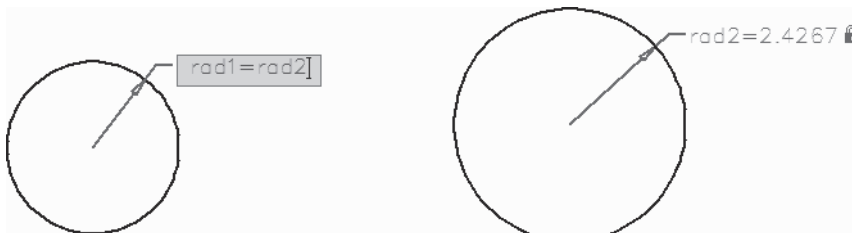
As you can see from this tutorial, the “constraint” word and the padlock icon are somewhat misleading, for both imply that dimensional constraints are fixed. They are not, for they can be edited both directly and through the new Parameters Manager palette.

TUTORIAL: LINKING DIMENSIONAL CONSTRAINTS

In the previous tutorial, you saw how dimensional constraints control the sizes of objects; constraints can also link sizes between objects. (Geometric constraints link the *position* of objects.)

You continue the tutorial from above by linking the radius of one circle with the other: when one changes, the other does too.

1. Double-click the text of the **rad1** circle.



2. Edit the text to read:
rad1=2*rad2
3. Press **Enter**. Notice that one circle is half the size of the other: circle **rad1** is twice the radius of second circle.
4. Edit the value of the second circle, and then press **Enter**. Notice that the first circle again changes size to accommodate the second one.
5. Move one of the circles. Notice that you can move it, but the radius (size) remains linked.



DELCONSTRAINT

The **DELCONSTRAINT** command erases dimensional constraints from drawings. (It also erases geometric constraints.)

Constraints cannot be removed with the **ERASE** command nor exploded with the **EXPLODE** or **XPLODE** commands. AutoCAD needs to keep track of parametric relationships, and so there is a separate command for dealing with undesired constraints.

TUTORIAL: ERASING CONSTRAINTS

1. To remove dimensional constraints from the drawing, start the **DELCONSTRAINT** command, as follows:
 - From the ribbon's Parametric tab, choose **Delete Constraints**.
 - Enter the **delconstraint** command, as follows:
Command: **delconstraint**
 - Alternatively, enter the **delcon** alias.
2. At the prompt, choose one of the radial dimensions from the previous tutorial:

All constraints will be removed from selected objects...

Select objects: *(Choose one of the radial dimensions.)*

Select objects: *(Press **Enter** to exit the command.)*

1 constraint removed

You can enter **All** at the 'Select objects' prompt to remove all constraints from the drawing. If you make a mistake, reverse it with the **U** command.



PARAMETERS

The **PARAMETERS** command creates links between dimensional constraints through the Parameters Manager palette.

Parameters are interactive formulas that control the size and position of objects. When you change the name of a parameter, AutoCAD automatically changes all related occurrences.

The Parameters Manager palette defines and edits parameters associated with dimensional constraints. Through it, AutoCAD maintains the relationships between dimensional constraints correctly. (There is no relationship between dimensional and geometric constraints.)

The palette also allows you to edit parameters. You can change names and expressions; values and types cannot, however, be edited.

TUTORIAL: EDITING PARAMETER VALUES

In this tutorial, you apply dimensional constraints to a rectangle, and then edit them through the Parameters Manager palette.

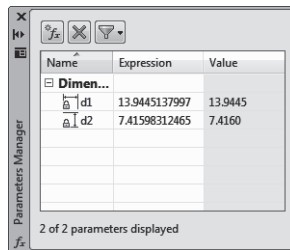
1. Begin this tutorial by drawing a rectangle with the **RECTANGLE** command.
2. Apply dimensional constraints to two sides of the rectangle with the ribbon's Parametrics tab: **Horizontal** for the horizontal one, and **Vertical** for vertical.



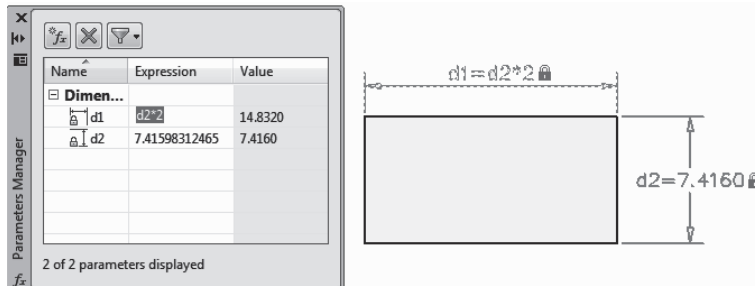
3. Open the Parameters Manager palette with the **PARAMETERS** command.
 - In the ribbon's Parametric tab, choose **Parameters Manager**.
 - Enter the **PARAMETERS** command:

Command: **parameters**

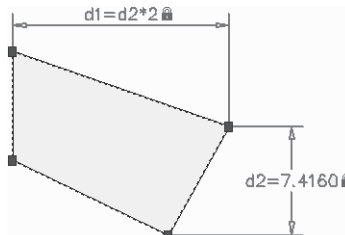
Notice the palette, in which the dimensional constraints appear as **d1** and **d2**.



4. Change the expression of **d1** to this formula: **d2*2**. It means that the width is always twice the height.



5. Select the rectangle, and then drag a grip to change the shape of the rectangle. Notice that the parameters update in the palette. This shows that the parameters in the palette and dimensional constraints are linked.



6. In the palette, change the value of the **d2** parameter to any arbitrary number. Notice that the rectangle changes its size automatically.

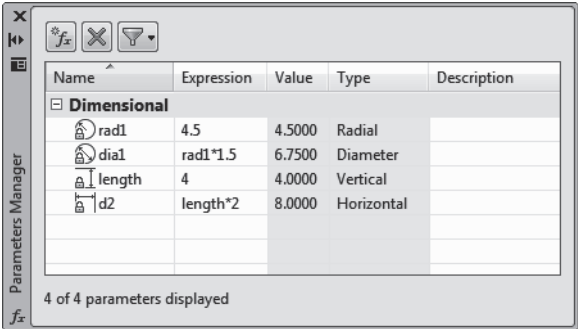
PARAMETERS MANAGER: ADDITIONAL METHODS

The `PARAMETERSSTATUS` system variable reports the status of the Parameters Manager palette: 0 means closed and 1 means open. To close the palette, click the x in the upper right corner, or use the `PARAMETERSCLOSE` command.

The `-PARAMETERS` command operates at the command line, creating new expressions, as well as editing, renaming, and deleting them.

Dialog Box

The Parameters Manager palette contains the following toolbar buttons, headers, and shortcut menu.



Toolbar Buttons

Add creates new user-defined expressions. The default names look like “user1” and can be renamed by you. You can enter numeric values (such as 4.5), expressions (such as `rad1*1.5`), or formulas (such as `length*2`). Do not use the equals sign (`=`). See the boxed text “Expressions in Parameters” for a list of all expressions supported by AutoCAD.

Remove deletes the selected expression.

Filter filters the display of parameters:

- **Display All Parameters** displays all parameters in the drawing, including unused ones.
- **Display Parameters Used In Expressions** displays only those parameters that are in use.

Headers

Click the name of a header to sort its content alphabetically; click again to sort in reverse order.

Name heads the list of variables names. Although AutoCAD assigns names automatically, you can give them names more meaningful than “d2” and “dia1” — such as Length and Diameter.

Expression specifies the parametric expression, and consists of constants, names, and expressions (formulae). See boxed text.

Value reports the value of the expression; you cannot edit this field.

Type reports the type of parameter (not user editable).

Description provides space to describe the parameter. Enter the purpose of the parameter to make it easier for others to follow your reasoning.

Shortcut Menu

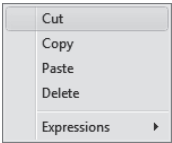
Double-click an expression, and then right-click to see the following shortcut menu:

Cut copies the expression to the Clipboard, and then deletes the expression.

Copy copies the expression to the Clipboard.

Paste pastes the contents from the Clipboard into the field.

Delete deletes the expression.



Expression provides a submenu of functions. In addition to those listed by the submenu, you can also use algebraic expressions, including +, -, *, and /. See boxed text.

EXPRESSIONS IN PARAMETERS

The following expressions can be used in parameters. To access a list of the expressions, double-click an Expression text entry box, and then right-click.

Expression	Meaning
+	Add
-	Subtract
/	Divide
*	Multiply
cos	Cosine
sin	Sine
tan	Tangent
acos	Arccosine
asin	Aresine
atan	Arctangent
cosh	Hyperbolic cosine
sinh	Hyperbolic sine
tanh	Hyperbolic tangent
atanh	Hyperbolic arctangent
sqrt	Square root
sign	Changes positive to negative and vice versa
exp	Exponent
floor	Rounds down
ceil	Ceiling (rounds up)
round	Rounds to the nearest integer, upwards or down
abs	Absolute value (makes negative numbers positive)
max	Maximum of two expressions
min	Minimum of two expressions
ln	Natural logarithm
log	Logarithm
pow	Raises one expression to the power of the second expression
exp10	Exponent to the power of 10
r2d	Converts radians to degrees
d2r	Converts degrees to radians
trunc	Truncates (removes decimal portion of number)
PI	3.141...
E	2.17...

Expression	Meaning
+	Add
-	Subtract
/	Divide
*	Multiply
cos	Cosine
sin	Sine
tan	Tangent
acos	Arccosine
asin	Aresine
atan	Arctangent
cosh	Hyperbolic cosine
sinh	Hyperbolic sine
tanh	Hyperbolic tangent
atanh	Hyperbolic arctangent
sqrt	Square root
sign	Changes positive to negative and vice versa
exp	Exponent
floor	Rounds down
ceil	Ceiling (rounds up)
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ln	Natural logarithm
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exp10	Exponent to the power of 10
r2d	Converts radians to degrees
d2r	Converts degrees to radians
trunc	Truncates (removes decimal portion of number)
PI	3.141...
E	2.17...

cos(expr)
sin(expr)
tan(expr)
acos(expr)
asin(expr)
atan(expr)
cosh(expr)
sinh(expr)
tanh(expr)
acosh(expr)
asinh(expr)
atanh(expr)
sqrt(expr)
sign(expr)
exp(expr)
floor(expr)
ceil(expr)
round(expr)
abs(expr)
max(expr1;expr2)
min(expr1;expr2)
ln(expr)
log(expr)
pow(expr1;expr2)
exp10(expr)
r2d(expr)
d2r(expr)
trunc(expr)
PI
E

Undocumented Expression

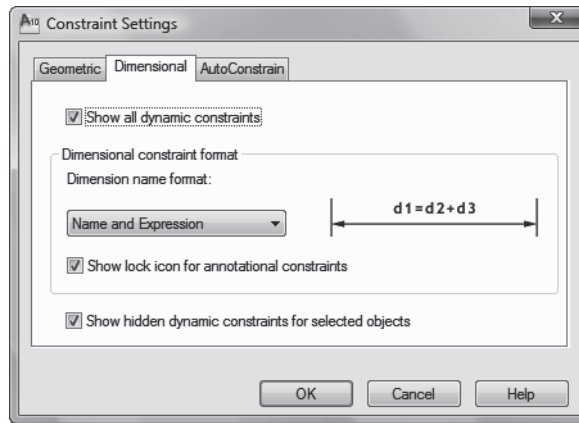
random Returns a random number between 0 and 1.



CONSTRAINTSETTINGS

The **CONSTRAINTSETTINGS** command specifies default conditions for dimensional and geometric constraints through a dialog box. In this chapter, we look at the dialog box's Dimensional tab of options; the next chapter covers the other two tabs.

1. To set options for dimensional constraints, start the **CONSTRAINTSETTINGS** command, as follows:
 - From the 2D ribbon's Parametric tab, click the  arrow.
 - Enter the **constraintsettings** command, as follows:
Command: **constraintsettings**
 - Alternatively, enter the **csettings** alias.
2. Click the Dimensional tab.

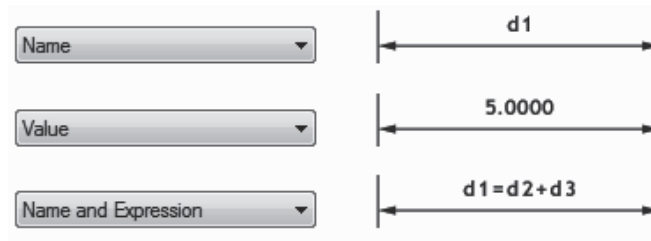


3. Change settings, and then click **OK**.

Dimensional Tab

Show All Dynamic Constraints toggles the display of dimensional constraints between invisible and visible. Dimensional constraints can clutter drawings, and so it can be useful to turn them off when not needed. (This option is stored in the **DYNCONSTRAINTDISPLAY** system variable.)

Dimension Name Format droplist displays (1) name only, (2) value only, or (3) name and expression, as illustrated below (**CONSTRAINTNAMEFORMAT**).



Show Lock Icon for Annotational Constraints toggles the display of the padlock icon (**DIMCONSTRAINTICON**).

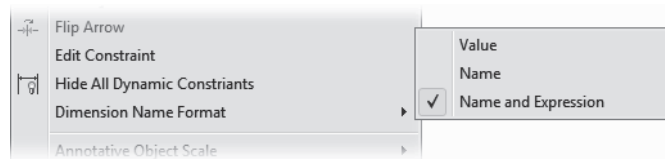
Show Hidden Dynamic Constraints for Selected Objects toggles the display of hidden constraints between displayed and not displayed when related objects are selected (**DYNCONSTRAINTMODE**).

CONSTRAINTS SETTINGS: ADDITIONAL METHODS

Some of the options in the Constraint Settings dialog box are controlled by the following system variables

- **DynConstraintDisplay** toggles the display of dimensional constraints.
- **DynConstraintIcon** toggles the display of the padlock icon.
- **DynConstraintMode** toggles the display of hidden dimensional constraints during selection.
- **ConstraintNameFormat** specifies the text format of dimensional constraints.
- **ConstraintRelax** toggles the enforcement of constraints during editing.
- **ConstraintSolveMode** toggles constraint behavior during editing.

Some of these system variables are also accessed through the context menu when a dimensional constraint is right-clicked.



DynConstraintDisplay

The **DynConstraintDisplay** system variable toggles the display of dimensional constraints. Clicking the **Show Dynamic Constraints** button on the ribbon's Parametric tab does the same thing, as does clicking **Hide All Dynamic Constraints** in the shortcut menu illustrated above.

DynConstraintDisplay	Meaning
0	Does not display dimensional constraints.
1	Displays dimensional constraints.

DimConstraintIcon

The **DimConstructionIcon** system variable determines when the padlock icon is displayed.

DimConstraintIcon	Meaning
0	Does not display the padlock icon.
1	Displays the padlock icon in dynamic constraints.
2	Displays the padlock icon in annotational constraints.
3 (default)	Displays the icon in both dynamic and annotational constraints.

DynConstraintMode

The **DynConstraintMode** system variable toggles the display of hidden dimensional constraints during selection. This affects objects you select whose constraints were hidden by **DynConstraintDisplay**.

DynConstraintMode	Meaning
0	Hidden constraints remain hidden when objects are selected.
1 (default)	Hidden constraints are displayed when objects are selected.

ConstraintNameFormat

The **ConstraintNameFormat** system variable specifies the text format of dimensional constraints.

ConstraintNameFormat	Meaning
0	Displays names, such as rad1.
1	Displays values, such as 3.141.
2 (default)	Displays expressions, such as rad1=3.141.

ConstraintRelax

The **ConstraintRelax** system variable reports whether constraints are enforced during editing. This sysvar is read-only.

ConstraintRelax	Meaning
0 (default)	Constraints are prevented from change (“enforced”).
1	Constraints can change (“relaxed”).

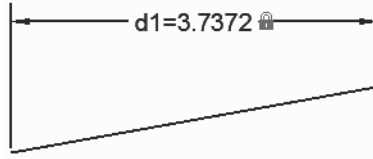
ConstraintSolveMode

The **ConstraintSolveMode** system variable toggles constraint behavior during editing.

ConstraintNameFormat	Meaning
0	Geometry can change when constraints are applied and modified.
1 (default)	Geometry is prevented from changing.

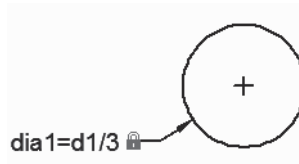
EXERCISES

1. Draw a line, and then apply a horizontal dimensional constraint to line **d1** — similar to the one illustrated below:

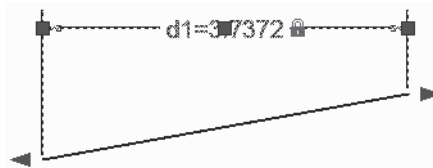


2. Continuing, draw a circle, and then use dimensional constraints to connect the sizes of objects, as follows:
 - a. Apply a diameter constraint to circle **dia1**.
 - b. Edit the circle's text to reference the line:

$$\text{dia1} = d1/3$$



- c. Write out the meaning of this formula:
3. Continuing, resize the objects by selecting its dimensional constraint, and then dragging a triangular handle.
 - a. Resize the line, and then describe what happens:



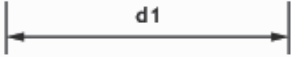
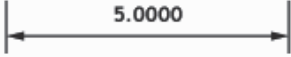
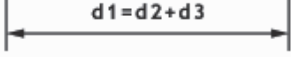
- b. Resize the circle, and then describe what happens.
4. Select the circle, and then drag one of its grips. Describe what happens:

CHAPTER REVIEW

1. What is the purpose of dimensional constraints?
2. List three ways in which dynamic dimensional constraints differ from regular dimensions:
 - a.
 - b.
 - c.
3. What does the padlock icon indicate?
What do parentheses around dimension text indicate?
4. Describe the differences between the following forms of dimensional constraints:
 - a. Dynamic
 - b. Annotational
 - c. Reference
 - d. Parametric
5. Can dimensional constraints switch between forms?
If so, how?
6. Can regular dimensions be converted to dimensional constraints?
If so, how?
7. Define “overconstrained.”
8. Describe what happens when you add a dimensional constraint to a fully constrained drawing.
9. Match the following abbreviations:

a. ang	i. distance
b. rad	ii. diameter
c. dia	iii. user-defined
d. d	iv. radius
e. user	v. angle
10. When objects are constrained, are they permanently fixed in place?
11. How can you unlock a constraint temporarily?
12. What happens when you hold down the **CTRL** key while dragging a grip of an constrained object?
13. Write out the formulas for the following constraint conditions:
 - a. Link the length of line 1 to the radius of arc 2.
 - b. Make the diameter of circle 3 half the length of line 4.
 - c. Make the length of line 5 four times the length of line 6.
 - d. Link the radius of circle 7 to the diameter of circle 8.
14. When entering formulas into the Parameters Manager palette, when would you use the equals (=) sign?
15. Name the ribbon tab that contains commands for placing dimensional constraints?
16. Can constraints be erased?
If so, how?

17. Match the text displayed by the dimensional constraint with the type of text:

- | | | |
|----|---|-------------------------|
| a. |  | i. Value |
| b. |  | ii. Name and expression |
| c. |  | iii. Name |

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CHAPTER 14

Using Geometric Constraints

Dimensional constraints control the sizes of objects, as you saw in the previous chapter. Related to them are geometric constraints, which control the position objects in drawings and the location of objects relative to each other.

In this chapter, you learn the following commands:

AUTOCONSTRAIN applies geometric constraints automatically.

GEOMCONSTRAINT applies specific geometric constraints to selected objects.

CONSTRAINTSETTINGS specifies default conditions for automatic and geometric constraints.

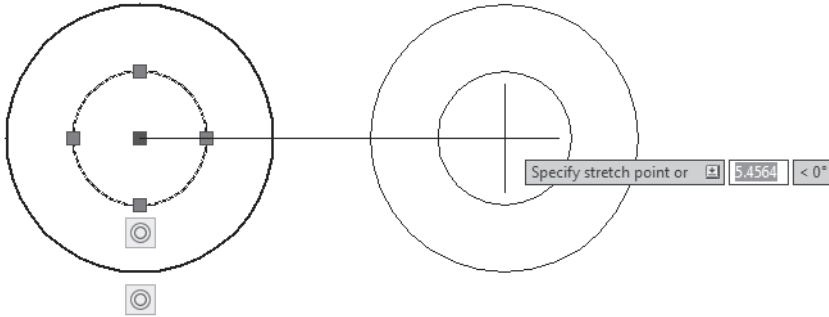
CONSTRAINTBAR toggles the visibility of constraint bars.

NEW TO AUTOCAD 2010 IN THIS CHAPTER

- All commands listed above are new to AutoCAD 2010.

ABOUT GEOMETRIC CONSTRAINTS

Geometric constraints keep objects together or in place. For example, when you apply the Fix constraint to an object, it is pinned to a fixed position in the drawing — like a pin in a piece of paper. Or when you apply the Concentric constraint to two circles, they remain concentric. As illustrated below, when the small circle is moved, and the larger one moves with it, because the constraint forces them to remain concentric.



Geometric constraints are like “sticky” object snaps. For instance, the Tangent osnap exists only for the split-second interval in which AutoCAD finds and then applies the tangency location; in contrast, the Tangency constraint forces lines to remember that they are tangent to arcs — even when the two do not touch, because they are connected through the constraint.

There are geometric constraints that match every object snap mode, such as Intersect and Perpendicular. As well, there are some extra constraints that have no osnap equivalent, such as Equal and Colinear. In total, AutoCAD provides these geometric constraints: Horizontal, Vertical, Perpendicular, Parallel, Tangent, Smooth, Coincident, Concentric, Colinear, Symmetric, Equal, and Fix. AutoCAD’s constraints operate primarily in 2D; programs like Autodesk’s Inventor also have 3D constraints. Spline frames cannot be constrained.

Objects do not need to touch for geometric constraints to work. For example, a line in a front view can have equal length to a line in the top view, so that changing one line changes the other; one end of the front-view line can always be vertical to the end top-view line. In this way, the views stay in step.

Geometric and dimensional constraints work together, and together they are known as *parametrics*. (Recall from the previous chapter that dimensional constraints define the sizes of geometric features.) Sometimes, the two types of constraints can replace each other. For example, the Perpendicular geometric constraint is the same as an Angular dimensional constraint set to an angle of 90 degrees.

AutoCAD does not allow you to add too many or conflicting constraints, known as *over constrained*. Normally, you want drawings to be *fully constrained*, which means there are no missing constraints.

While individual constraints can be applied with the GEOMCONSTRAINT command, it is easier and faster to let the AUTOCONSTRAIN command to find all possible geometric constraints for you, after which you remove and add individual ones as necessary. Once the geometric constraints are in place, you typically add the dimensional constraints described in the previous chapter.



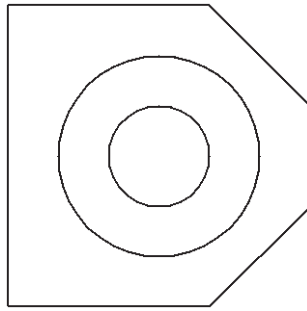
AUTOCONSTRAIN

The **AUTOCONSTRAIN** command automatically constrains selected objects. It finds probable constraints between geometric objects that are within the tolerance distance specified by the Constraint Settings dialog box. (This dialog box also controls *which* constraints are applied, and in which order of priority.)

TUTORIAL: CONSTRAINING GEOMETRY AUTOMATICALLY

In this tutorial, you open an existing drawing, and then automatically constrain all geometry.

1. From the Companion CD, open the *geom-constraint.dwg* file, a drawing of a part made of lines and circles.



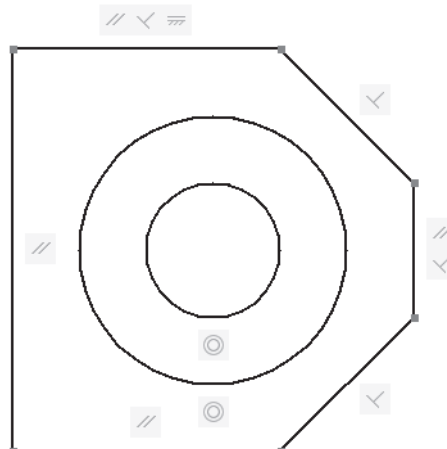
2. To apply geometric constraints to the entire drawing, start the **AUTOCONSTRAIN** command, as follows:
 - In the ribbon's Parametrics tab, choose the **Auto Constrain** button.
 - Enter the **autoconstrain** command:

Command: **autoconstrain**

3. At the prompt, enter 'All' to select all objects:
 Select objects or [Settings] <Settings>: (Enter **all** to select all objects.)
 Select objects or [Settings] <Settings>: (Press **Enter**.)



Note You don't have to select everything in the drawing; you can select specific objects using any selection mode. This command's **Settings** option displays the Constraint Settings dialog box's AutoConstrain tab, as described later.



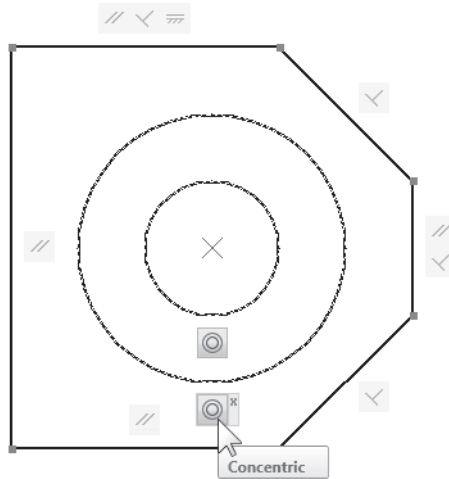
Notice the many toolbar-like “tags” that appear next to the lines and circles. These are called “constraint tags”; their icons report the constraint applied to adjacent geometry.



Note Constraint bars can be turned off with the **CONSTRAINTBAR** command, or through buttons on the ribbon’s Parametric tab.

There are many constraints in the drawing! (I count 11.) All of them connect at least two objects, and so it becomes difficult to determine the connections, even in a drawing as simple as this one. For this reason, AutoCAD lets you view objects related to constraints in two ways: by constraint and by object.

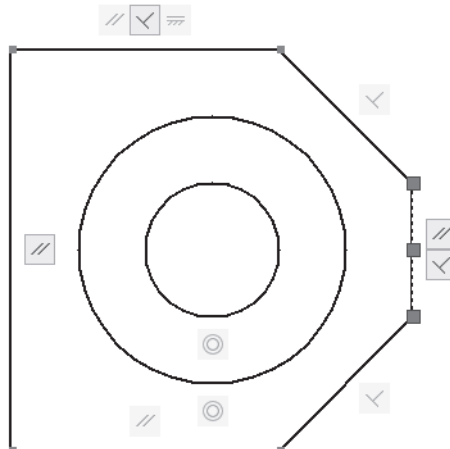
4. First, view constrained objects by constraint. Click on one of the Concentric tags, as shown below.



Notice that AutoCAD provides you with a bunch of information:

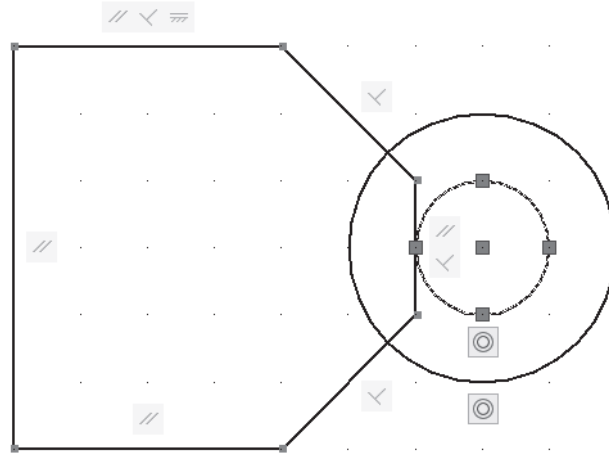
- Two concentric tags become opaque to help them stand out from other tags. There are two tags and they relate to one another, because the inner circle is concentric with the outer one, and vice versa.
- The two circles constrained by concentricity are highlighted.
- The center point of concentricity is indicated by the X.

5. Alternatively, you can view connected objects by object. Select the short vertical line.



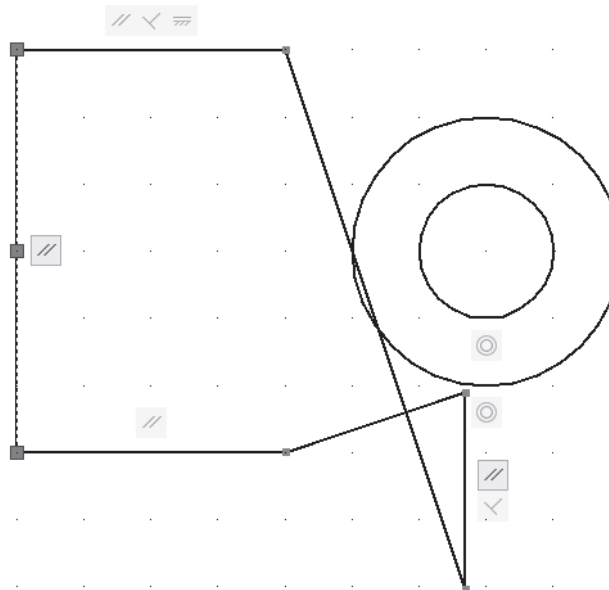
Notice that AutoCAD highlights the related constraints:

- The perpendicular constraint on the top line.
 - The parallel constraint on the long vertical line.
- To view the remaining connections, click on other objects and other constraint bars.
 - Prove to yourself that the constraints hold the otherwise-individualistic circles together: try moving them by dragging one of their grips. Notice that the two circles move as a whole, independent of the polygons — because the polygon and circles are not constrained to each other, nor are they fixed in place. (See Exercises at the end of this chapter.)



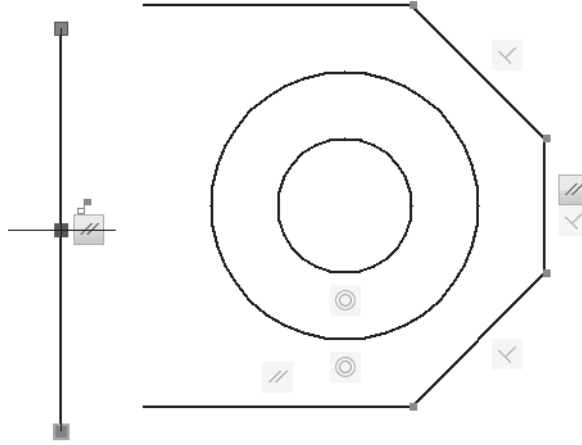
Moving one circle moves the other.

- When you attempt to move one of the lines making up the polygon, it stays connected to the others, but the polygon may distort, depending on how constraints are applied.



Moving one line also moves all connected lines, sometimes with unexpected results.

9. To unlock constraints temporarily, follow these steps:
 - a. Select an object, such as one of the lines.
 - b. Click a grip so that it turns red.
 - c. Tap the **CTRL** key (don't hold it down*).
 - d. Drag the grip, and notice that the line separates from the polygon.

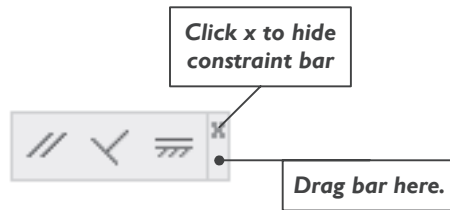


*) AutoCAD reacts differently when you hold down the **CTRL** key while dragging the grip: a copy of the object is made, complete with constraint(s).

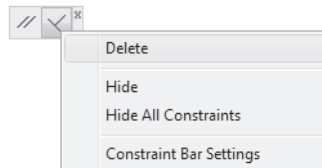
TUTORIAL: MODIFYING GEOMETRIC CONSTRAINTS

After AutoCAD adds constraints to drawings automatically, you may wish to modify them. There isn't, however, much modification available, short of hiding, deleting, or adding new constraints. Unlike dimensional constraints, geometric constraints cannot be edited — because there is no need to.

1. To hide all constraint bars, click the **Hide All** button found in the ribbon's Parametric tab. (Alternatively, you can enter the **CONSTRAINTBAR** command, and then specify the **Hideall** option; the **Showall** option displays them all again.)
2. To hide an individual constraint bar, click the **x** in its upper right corner.



3. To move a constraint bar, drag it around by its tab. (See figure above.)
4. To delete an individual constraint from the bar, right-click it, and then choose **Delete** from the shortcut menu.



5. To delete all constraints associated with an object, use the **DELCONSTRAINT** command. See the previous chapter.
6. To add more constraints, see the **GEOMCONSTRAINT** command described below.



Notes The **DELCONSTRAINT** command erases all geometric and dimensional constraints from a selection set of objects — both visible and hidden.

The Parameters Manager palette (**PARAMETERS** command) does not apply to geometric constraints.



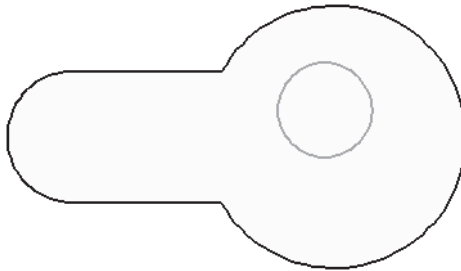
GEOMCONSTRAINT

The **GEOMCONSTRAINT** command applies geometric relationships between objects or to points on objects.

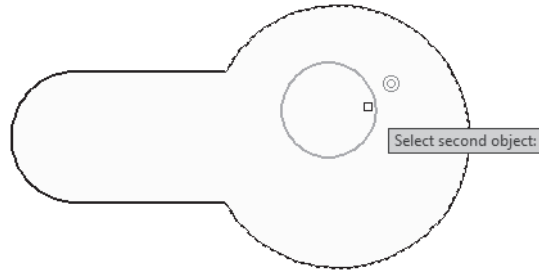
TUTORIAL: APPLYING CONSTRAINTS MANUALLY

In this tutorial, you force the circle to remain centered in the keyhole shape with the **GEOMCONSTRAINT** command's **Concentric** option.

1. Start AutoCAD with a new drawing, and then draw a keyhole shape with the **PLINE** command. See figure below.
2. Add a circle with the **CIRCLE** command; its location in the drawing is unimportant.

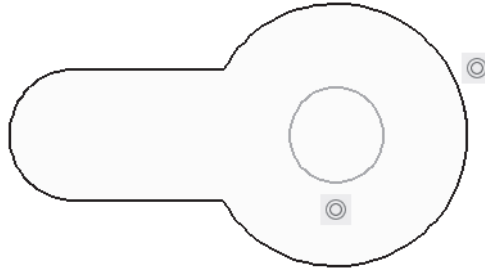


3. Start the **GEOMCONSTRAINT** command, as follows:
 - In the ribbon's Parametrics tab, choose the **Concentric** button.
 - Enter the **geomconstraint** command:
Command: **geomconstraint**
 - Alternatively, enter the **gcon** alias.
4. If necessary, specify the **Concentric** option:
Enter constraint type
[Horizontal/Vertical/Perpendicular/PArallel/Tangent/SMooth/COincident/CONcentric/
COLinear/Symmetric/Equal/Fix]<Coincident>: (Type **CON**.)
5. AutoCAD prompts you to select an object:
Select first object: (Choose the polyarc.)
Choose the large polyarc at the right end of the polyline.
6. At the next prompt, choose the circle.
Select second object: (Choose the circle.)



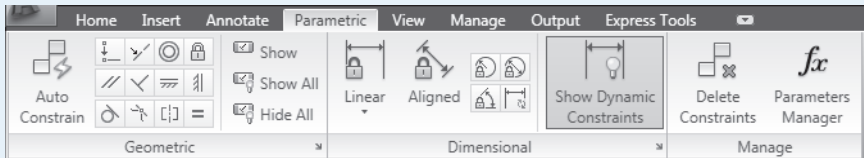
Located near the cursor, the blue icon reminds you that concentric mode is current.

Notice that AutoCAD displays constraint bars to indicate the type and approximate location of the applied constraint.



USING THE RIBBON WITH CONSTRAINTS

Autodesk has designed the **GEOMCONSTRAINT** command so that it is easier to use with the ribbon than with command prompts. Geometric constraints are found in the Parametric tab's Geometric panel:



GEOMETRIC PANEL

 **Auto Constrain** applies geometric constraints automatically to the entire drawing, or to selected objects. (This button activates the **AUTOCONSTRAIN** command.)

For the meaning of individual geometric constraints, see Geometric Constraint Catalog.

 **Show** displays constraint tags, except hidden ones (**CONSTRAINTBAR** command).

 **Show All** displays constraint tags, including hidden ones (**CONSTRAINTBAR** command).

 **Hide All** hides all constraint tags from view (**CONSTRAINTBAR** command).

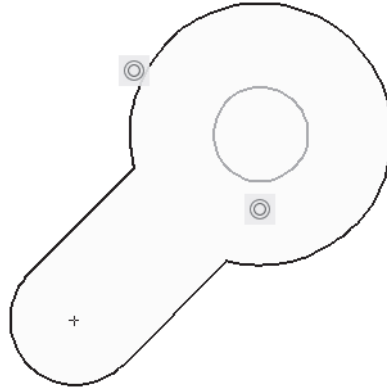
Click the  arrow to access the Constraint Settings dialog box; see the **CONSTRAINTSETTING** command.

MANAGE PANEL

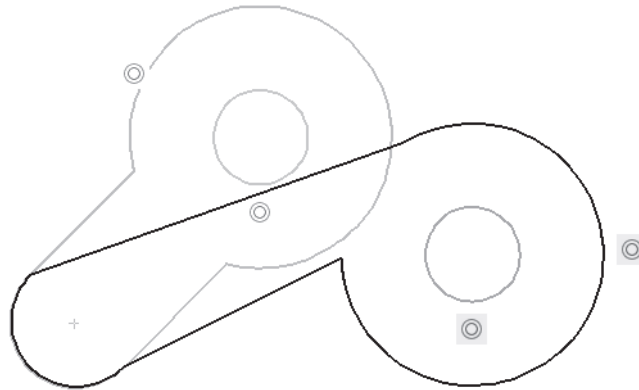
 **Delete Constraints** removes constraints from all or selected objects (**DELCONSTRAINTS** command).

(Parameters Manager applies to dimensional constraints only. Buttons in the Dimensional panel relate to dimensional constraints.)

7. With the constraint in place, let's see what happens when the objects are moved. First, use the **ROTATE** command to rotate the keyhole about its left end (the base point is centered on the small polyarc). Notice that the circle moves with the keyhole due to the concentric constraint.



8. Drag the circle. Notice that the keyhole stretches itself to keep the large polyarc centered around the circle.



Geometric Constraint Catalog

AutoCAD provides many types of geometric constraints. Below, they are listed in alphabetical order.

Be aware that some constraints act differently, depending on the option you select. For example, the Fix constraint allows the selected object to be scaled and rotated about the lock point when you use the 'Select point' option. But when you select the same object with the 'Object' option, it is truly fixed in place: it cannot be scaled, rotated, or moved.

Coincident

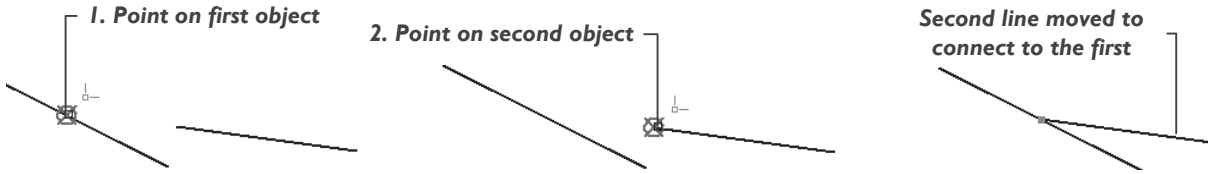
The **Coincident** constraint forces a point of the second object to lock to a point of the first. Coincident points can be endpoints, midpoints, or any other point you pick; objects can be lines, polylines and polyarcs, circles, arcs, ellipses, and splines.

Select first point or [Object/Autoconstrain] <Object>: *(Pick a constraint point, or enter an option.)*

Select second point or [Object]<Object>: *(Pick another constraint point or type O.)*

First point specifies the endpoint of the first object.

Second point specifies the endpoint of the second object, which will be made coincident to the first one, as illustrated below.



Autoconstrain constrains multiple objects automatically, in the manner of the **AUTOCONSTRAIN** command.

Object selects an object instead of the constraint point; works with lines, polylines and polyarcs, circles, arcs, ellipses, and splines. Prompts you:

Select object: *(Choose an object.)*

Select point or [Multiple]: *(Pick a point on the object, or type **M**.)*

Multiple selects more than one additional point to coincide with the first.



Note Use the **Object** option to slide a circle or polygon along a line by constraining the center of the circle to the line.



COLinear

The **COLinear** constraint forces one object to lie in the same line as a second; works with lines and polyline segments. It displays the same prompts as Equal.



CONcentric

The **CONcentric** constraint forces two curved objects to share the same center point; works with circles, arcs, polyarcs, and ellipses. Displays the same prompts as Perpendicular:

Select first object: *(Choose the object to be constrained.)*

Select second object: *(Choose the reference object.)*

= Equal

The **Equal** constraint forces one line or polyline segment to be the same length as a second. It also forces one arc or circle to have the same radius as the second. This constraint also works with polyarcs.

Select first object or [Multiple]: *(Choose an object, or type **M**.)*

Select second object: *(Choose a second object.)*

First object chooses a line, polyline, polyarc, arc, or circle.

Second object chooses another object, and then scales it to be the same size as the first.

Multiple selects additional objects; repeats the 'Select object to make equal to the first' prompt until you press Enter.



Fix

The **Fix** constraint locks objects in position. This constraint works with all objects.

Select point or [Object] <Object>: *(Pick a point on an object, or type **O**.)*

Select point allows the object to be scaled (resized) and rotated about the lock point.

Object locks the object in place; it cannot be resized, rotated, or moved.

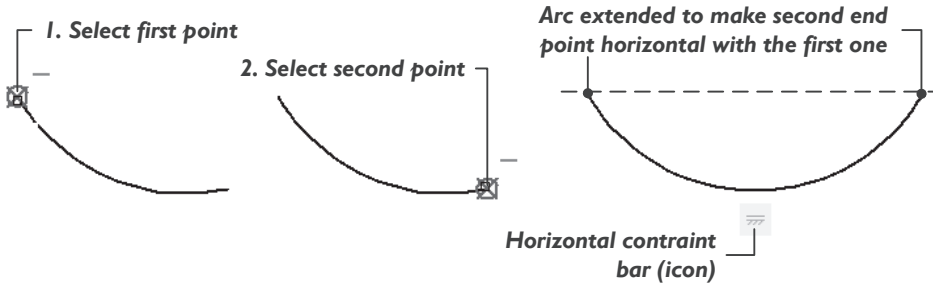
Horizontal

The **Horizontal** constraint forces geometry to be horizontal:

Select an object or [2Points]: (*Choose an object, or type 2P.*)

Object option forces selected object to be horizontal. If the object is an arc, then its endpoints are extended to make the two pick points horizontal.

2Points option picks two points on an object; the orientation of the two points is made horizontal, as illustrated below. This is useful for orienting an arc or polyline. Works with lines, polyline segments, arcs and circles.



Parallel

The **Parallel** constraint forces one object to be parallel to a second. The first is rotated to be parallel to the second.

Select first object: (*Choose the object to be constrained.*)

Select second object: (*Choose the reference object.*)

First object chooses a line or a polyline segment to be made parallel.

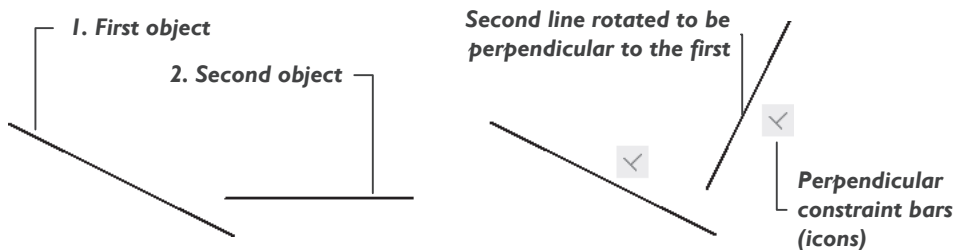
Second object picks the reference line or polyline segment; the second object is rotated to be parallel to the first.

Perpendicular

The **Perpendicular** constraint forces a first object to be perpendicular to a second. The second is rotated to be perpendicular to the first, as illustrated below.

Select first object: (*Choose the object to be constrained.*)

Select second object: (*Choose the reference object.*)



Smooth

The **Smooth** constraint forces the connection between a spline and another open curve to be smooth; endpoints become coincident. The first object must be a spline; the second object can be another spline, an arc, or a polyarc.

Select first spline curve: (*Choose a spline.*)

Select second curve: (*Choose another spline, arc, or polyarc.*)

First curve must be a spline.

Second curve can be another spline, an arc, or a polyarc.



Note AutoCAD applies a G2 curvature continuous condition with the SMOOTH constraint. *G2 curvature* specifies that the second derivative of the curves is equal at the point of contact. This ensures that the radius of curvature is continuous across the spline and arc or polyarc. The “G” is short for geometry; there are also G1 and G3 curves that apply the first (tangent) and third derivatives across curves and surfaces. G2 is the practical maximum in CAD.

Symmetric

The **Symmetric** constraint forces two objects to have mirror-like symmetry about a line of symmetry. Depending on which option is selected, the object selected second may be scaled to be the same size as the first.

Select first object or [2Points] <2Points>: *(Choose an object, or type 2P.)*

Select second object: *(Choose another object.)*

Select symmetry line: *(Pick an object.)*

First object chooses one object; works with lines, polylines and polyarcs, circles, arcs, and ellipses.

Second object chooses the second object, and then scales it to be a mirror image of the first.

2Points chooses pick points on two objects; the objects maintain their size.

Symmetry line chooses an object to use as the mirror (aka symmetry) line.



Note The symmetry constraint makes the angles of lines symmetrical; in arcs and circles, the center points and radii are made symmetrical. Polylines and other objects may become distorted to match the constraint conditions.

Tangent

The **Tangent** constraint forces one straight or curved object to be tangent to a curved object. Objects need not touch one another. This option works with lines, polylines and polyarcs, circles, arcs, and ellipses. Displays the same prompts as Perpendicular:

Select first object: *(Choose the object to be constrained.)*

Select second object: *(Choose the reference object.)*

Vertical

The Vertical constraint forces geometry to be vertical; displays the same prompt as Horizontal:

Select an object or [2Points]: *(Choose an object, or type 2P.)*



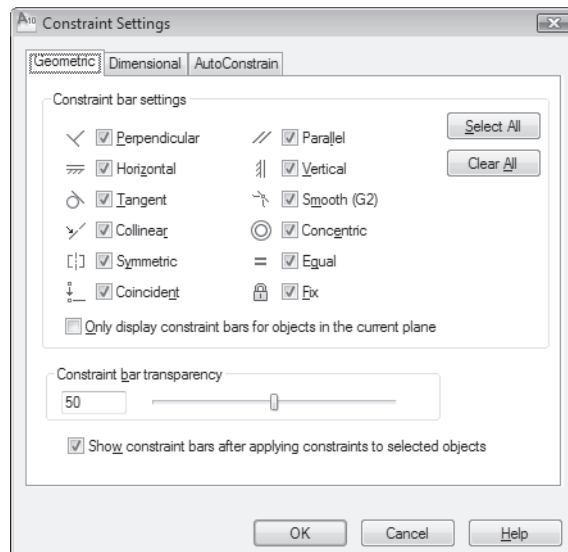
CONSTRAINTSETTINGS

The **CONSTRAINTSETTINGS** command specifies default conditions for geometric and dimensional constraints through a dialog box. In this chapter, we look at the dialog box's Geometric and AutoConstrain tabs; the previous chapter covered the other tab.

1. To set options for dimensional constraints, start the **CONSTRAINTSETTINGS** command, as follows:
 - From the 2D ribbon's Parametric tab, click the  arrow.
 - Enter the **constraintsettings** command, as follows:
Command: **constraintsettings**
 - Alternatively, enter the **csettings** alias.
2. Choose the Geometric or AutoConstrain tab.
3. Change settings, and then click **OK**.

Geometric Tab

The Geometric tab controls how geometric constraints are added to objects in drawings.



Constraint Bar Settings toggles the display of individual geometric constraints. (These settings are stored in system variable **CONSTRAINTBARMODE**.)

Only Display Constraint Bars for Object in the Current Plane toggles the display of constraints (1) always or (2) only when in the current UCS.

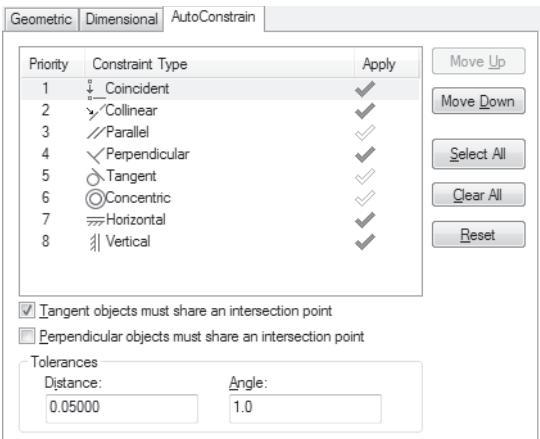
Select All turns on all geometric constraints; **Clear All** turns off all geometric constraints.

Constraint Bar Transparency adjusts the translucency of constraint bars from 10 (almost opaque) to 90, almost transparent (**CBARTRANSPARENCY**).

Show Constraint Bars After Applying Constraints to Selected Objects toggles the default visibility of constraint bars between invisible and visible (**CONSTRAINTBARDISPLAY**).

AutoConstrain Tab

The AutoConstrain tab controls how geometric constraints are added automatically with the AUTOCONSTRAIN command.



Priority specifies the order in which constraints are applied automatically with the AUTOCONSTRAIN command; use the **Move Up** and **Move Down** buttons to change the order. The checkmarks in the **Apply** column toggle individual geometric constraints.

Select All sets all geometric constraints as autoconstraints; **Clear All** turns off autoconstraining for all geometric constraints. **Reset** returns the AutoConstrain settings to default values.

Tangent Objects Must Share an Intersection Point determines how objects must relate when applying the Tangent constraint:

- On: Tangent constraints are applied only when objects touch.
- Off: Tangent constraints are applied if objects would touch after being theoretically extended.

Perpendicular Objects Must Share an Intersection Point determines how two objects relate for applying the Perpendicular constraint:

- On: Objects must touch.
- Off: Objects needn't touch.

Tolerances Distance specifies the distance within which the constraints are applied automatically by the AUTOCONSTRAIN command.

Tolerances Angle specifies the angle within which angle-related constraints are applied automatically.

CONSTRAINTS SETTINGS: ADDITIONAL METHODS

Some of the options in the Constraint Settings dialog box are controlled by the following system variables.

ConstraintBarDisplay

The **ConstraintBarDisplay** system variable toggles the display of all constraint bars: (0) off, and (1) on.

CBarTransparency

The **CBarTransparency** system variable stores the current translucency of constraint bars; range is 10 to 90 per cent.

ConstraintRelax

The **ConstraintRelax** system variable indicates whether constraints are (0) enforced or (1) relaxed when editing objects.

ConstraintSolveMode

The **ConstraintSolveMode** system variable determines the behavior of geometry when applying or editing constraints.

ConstraintSolveMode	Meaning
0	Allows geometry to change when constraints are applied or modified.
1	Prevents geometry from changing.

ConstraintBarMode

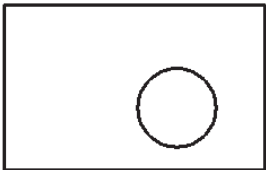
The **ConstraintBarMode** system variable determines which geometrical constraint icons are displayed on constraint bars. The default value is 4031, meaning all are displayed.

ConstraintBarMode	Meaning	ConstraintBarMode	Meaning
1	Horizontal	128	Concentric
2	Vertical	256	Colinear
4	Perpendicular	512	Symmetric
8	Parallel	1024	Equal
16	Tangent	2048	Fix
32	Smooth	4031 (default)	All displayed
64	Coincident		

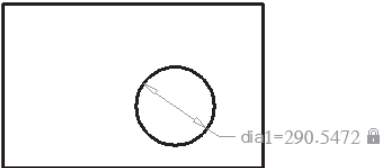
TUTORIAL: COMBINING DIMENSIONAL AND GEOMETRIC CONSTRAINTS

In this tutorial, you use dimensional and geometric constraints to force a rectangle to be twice the size of a circle, and the circle to be at the center of the rectangle.

1.
- Create the simple drawing of a rectangle (**RECTANGLE** command) and circle (**CIRCLE** command), as shown below. The size of the rectangle and the circle's location within it are unimportant.



2.
- From the Parametric tab's Dimensional panel, choose **Diameter**.
Select arc or circle: *(Choose the circle.)*
Dimension text = 290. Specify dimension line location: *(Pick a point.)*



(The value displayed may differ in your drawing.)

3.
- Force the length of the sides of the rectangle to be twice the diameter of the circle. From the Parametric tab's Dimensional panel, choose **Horizontal**. (Do not confuse this dimensional constraint with the Horizontal dimensional constraint.)

Select first constraint point or [Object]: *(Pick an endpoint of the rectangle. Note that object snap is not necessary, because AutoCAD automatically snaps to the end point.)*

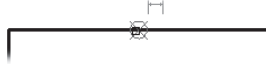


Notice that AutoCAD highlights the endpoint with a red icon.

4. For the second point of the dimensional constraint:

- a. Choose the midpoint:

Select second constraint point: *(Pick the midpoint of the rectangle.)*



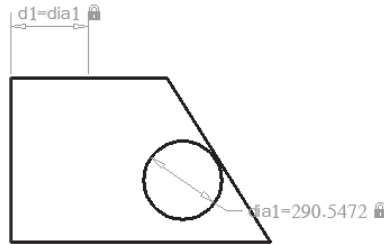
- b. Locate the dimension line:

Specify dimension line location: *(Pick a point.)*

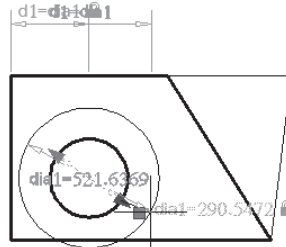


- c. Edit the dimension text to read “d1 = dia1.”

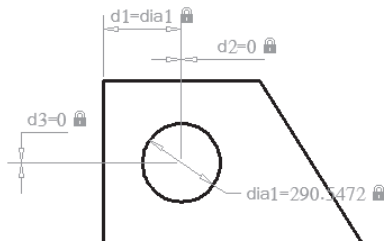
The length of half of the line is now constrained to be equal to the diameter of the circle. Notice that the rectangle changes shape immediately to meet the demand of the dimensional constraint.



5. Pick the **dia1** diameter dimension. Notice the two triangle grips. Drag one of the grips. Not only does the circle change diameter, but the polygon (no longer a rectangle) also changes size and shape.



6. To force the circle to the center of the rectangle, add a horizontal and a vertical dimension, each of 0 units.

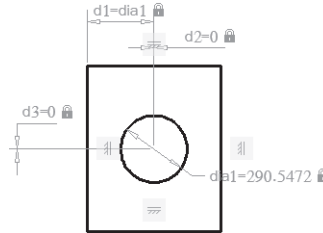


Attach dimensional constraints to the midpoints of lines and the center of the circle, as illustrated above.

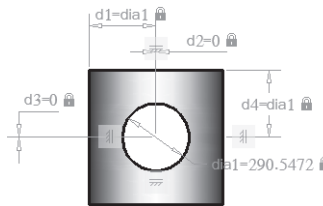
7. To force the polygon to remain rectangular, choose **Horizontal** from the Geometric panel, and then apply it to the horizontal lines of the polygon.

Select an object or [2Points]: (Pick a line.)

Repeat, applying **Vertical** geometric constraints the vertical sides. As you apply the geometric constraints, notice that the polygon straightens out.

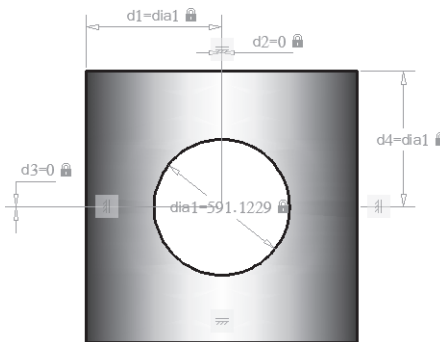


8. Drag the diameter constraint's arrow grips. As the circle and rectangle change size, notice that the rectangle remains rectangular.
9. Finish the job by adding Vertical dimensional constraints to keep the circle centered vertically.

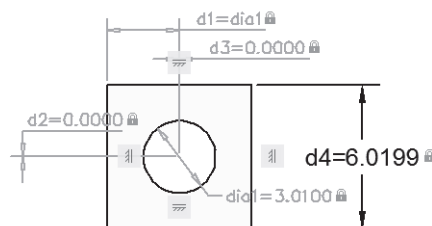
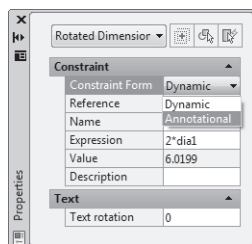


To see that hatches also obey constraints, fill the rectangle with a gradient hatch pattern (**GRADIENT** command).

10. When you drag the diameter constraint's triangle grips, all three elements react: the circle, the square, and the gradient fill. The circle remains centered, the square remains square, and the gradient fill stays filled.



11. The dimensional constraints are in dynamic form; for plotting, you should change them to annotational form, as follows:

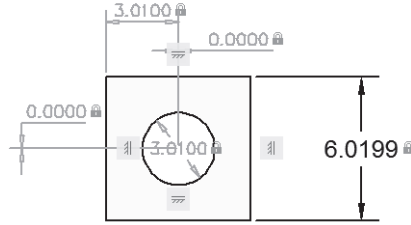
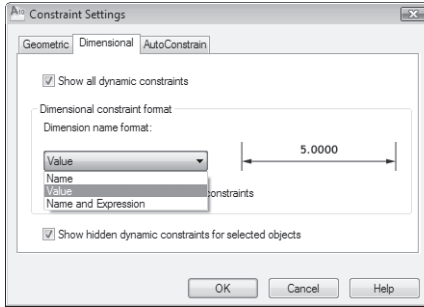


- Start the **PROPERTIES** command.
- Select the vertical dimension on the right.

- c. In the Constraint Form droplist, choose **Annotational**.

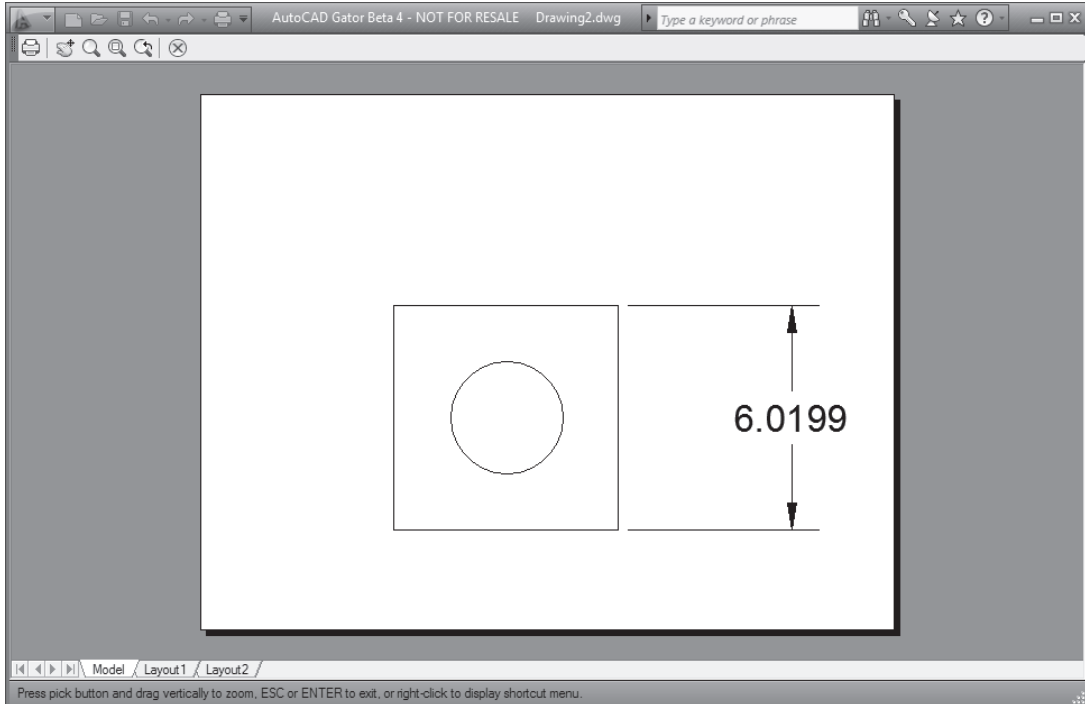
Notice that the dimension text and lines turn black, the sign that the constraints are annotational.

12. The annotational dimension displays the dimension as a formula. To display it as a value, use the Constraint Settings dialog box:
 - a. Start the **CONSTRAINTSETTINGS** command.
 - b. Choose the **Dimensional** tab.
 - c. In the Dimension Name Format droplist, choose **Value**.
 - d. Click **OK**.



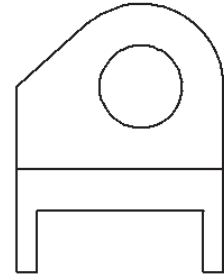
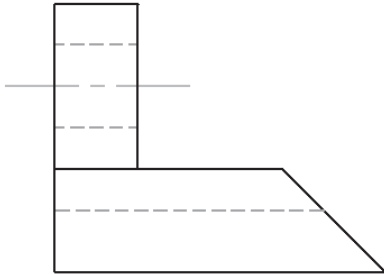
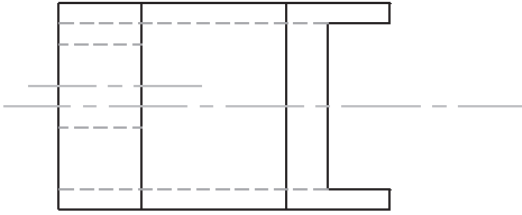
Notice that the dimensional constraints now display their values only.

15. "Plot" the drawing with the **PREVIEW** command to see that only annotational dimensions are not.

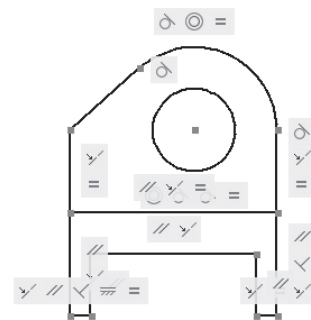
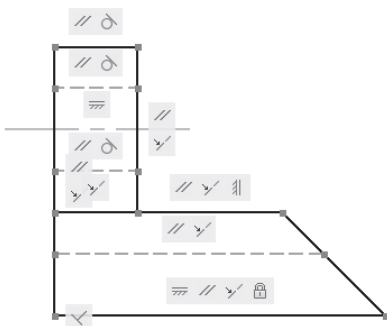
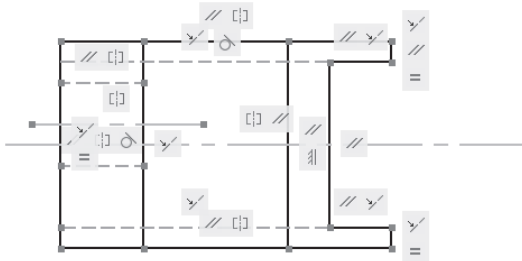


EXERCISES

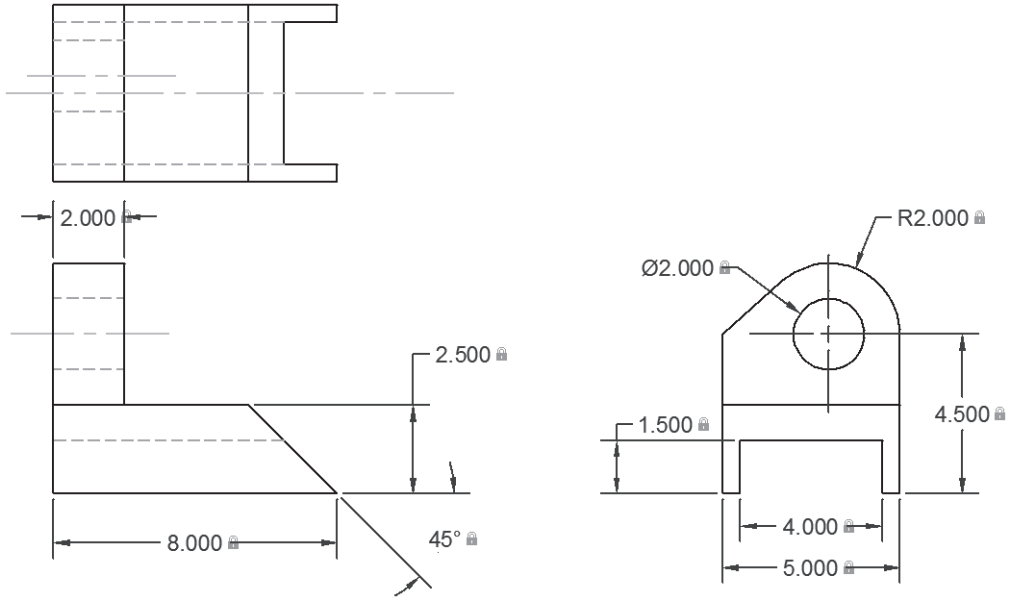
- I. From the Companion CD, open the *parametrics-2.dwg* file.



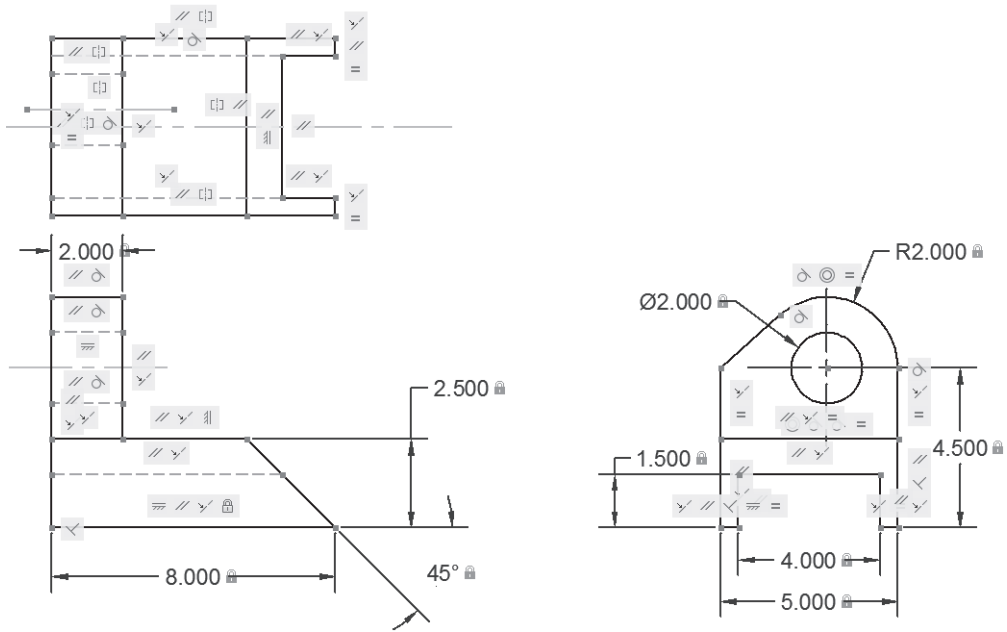
- a. Apply geometric constraints, as illustrated below. Hints:
- Apply geometric constraints first, and then apply dimensional ones.
 - Use AutoConstrain in small batches, otherwise you may have unintended results, such as diagonal lines forced perpendicular.
 - Initially accept all default values for dimensional constraints, and then edit them afterwards.



- b. Apply dimensional constraints, as illustrated below.



Your completed drawing should look similar to the figure.



2. In this exercise, you continue from the “Constraining Geometry Automatically” tutorial, found at the beginning of this chapter.
 - a. Open the *geom-constraint.dwg* file from the Companion CD.
 - b. Apply the **AUTOCONSTRAIN** command.
 - c. Recall that you were able to move the circles out of the polygon. What kind of a constraint would keep the circles positioned in the polygon?
Apply the constraint to the drawing.
 - d. What kind of constraint keeps the drawing from moving?
Apply the constraint to the drawing.

CHAPTER REVIEW

1. What is the purpose of geometric constraints?
2. How do geometric constraints differ from object snaps?
3. When are drawings full constrained?
4. Describe the primary difference between the **AUTOCONSTRAIN** and **GEOMCONSTRAINT** command.
5. Define *tolerance distance*.
6. What are *constraint tabs*?
7. List two ways that you can identify which constraints and objects are connected with each other:
 - a.
 - b.
8. Describe the function of the following constraints:
 - a. CONcentric
 - b. Equal
 - c. Parallel
 - d. Symmetric
9. Must objects touch to be constrained to each other?
10. Indicate the difference between using the Fix constraint's 'Select point' option and 'Object' option:
 - a. Select point
 - b. Object
11. What is the effect of the Horizontal constraint's '2Points' option on arcs?
12. Which object cannot be constrained?
13. How might you turn off the display of constraint bars?
14. Which type of constraint should be applied to drawings first — dimensional or geometric?

Bonus Question

15. The SMOOTH constraint applies G2 curvature between curves. Define the parts of the name "G2":
 - a. G
 - b. 2

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CHAPTER 15

Reporting on Drawings

A benefit of drawing with CAD is that you can obtain information from the drawing. AutoCAD's inquiry commands provide the following kinds of information:

ID reports the x, y, z coordinates of single points.

MEASUREGEOM reports distances, angles, areas, and volumes.

MASSPROP reports the area, perimeter, centroid, and so on of regions and solids.

LIST and **PROPERTIES** report information about objects.

TABLE creates tables of data.

TABLEEDIT edits data in tables.

TABLESTYLE specifies table styles.

DWGPLOTS reports and stores information about the drawing.

TIME reports on time spent editing the drawing.

SETVAR reports the settings of system variables.



Note AutoCAD always works to 14 digits. All reporting commands display accuracy and units according to the current setting of the **UNITS COMMAND**; sometimes, the actual values are not returned. For instance, the value 3.128362563 is stored by AutoCAD, but could be displayed as 3.13, $3\frac{1}{8}$, or 3 — depending on the setting of **UNITS**.

NEW TO AUTOCAD 2010 IN THIS CHAPTER

- AutoCAD has new system variables, as well as new values for existing sysvars, as detailed in Appendix C.
- The new **MEASUREGEOM** command reports distances, areas, angles, and volumes.



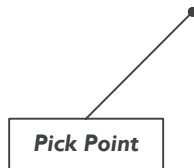
The **ID** command reports the x, y, z coordinates of single points. The command is transparent, so it can be used during other commands.

TUTORIAL: REPORTING POINT COORDINATES

1. To find the coordinates of points, start the **ID** command:
 - In the ribbon's Home tab, choose the **ID Point** button in the Utilities panel.
 - At the 'Command:' prompt, enter the **id** command.

Command: **id** (Press ENTER.)

2. In all cases, AutoCAD prompts you to pick a point in the drawing.
Specify point: (Pick a point.)



AutoCAD reports the x, y, z coordinates of the point in the command prompt bar:

X = 13.3006 Y = 4.4596 Z = 0.0000

The “z coordinate” is actually the elevation, and will always report 0.0000 unless changed by the **ELEV** command.

To capture the point of a geometric feature accurately, use an object snap mode, such as **ENDpoint** or **CENTER**.

LastPoint and @

AutoCAD stores the x, y, z-coordinate in the **LASTPOINT** system variable, which you can access in the command that follows by entering **@** at the prompt. Here is an example using the **LINE** command:

Command: **line**

Specify first point: **@**

Specify next point or [Undo]: (Pick a point.)

And an example using the **CIRCLE** command:

Command: **circle**

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: **@**

Specify radius of circle or [Diameter]: (Pick a point.)

The coordinates are stored in **LASTPOINT** only until the next 'Specify point' prompt, however. As soon as you pick another point, or enter coordinates at the keyboard, AutoCAD stores the new coordinates in the **LASTPOINT** system variable.

MEASUREGEOM

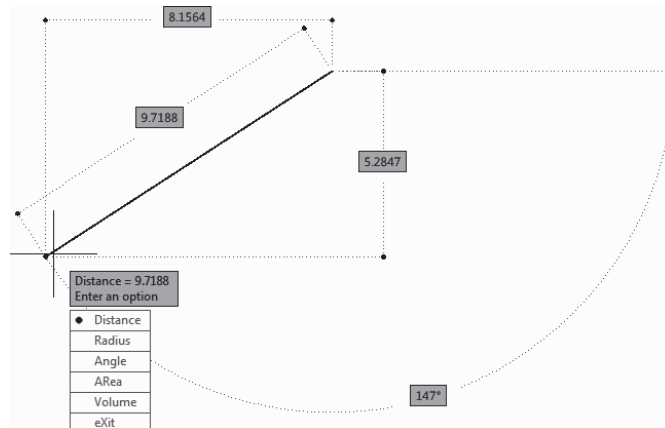
The MEASUREGEOM command reports distances, angles, radii, areas, and volumes between points and of objects.

TUTORIAL: REPORTING DISTANCE AND ANGLE MEASUREMENTS

In this tutorial, you draw several objects, and then measure them with the MEASUREGEOM command.

1. Start a new drawing with the *acad.dwt* template file, and then draw a line at an angle. It can be of any length.
2. To ensure that measurements are accurate, turn on INTersection object snap.
3. To find the length and angle, start the **MEASUREGEOM** command:
 - In the ribbon's Home tab, choose the **Measure** button in the Utilities panel.
 - At the 'Command:' prompt, enter the **measuregeom** command:
Command: **measuregeom** (Press ENTER.)
 - Alternatively, enter the **mea** alias at the 'Command:' prompt.
4. In all cases, AutoCAD prompts you to choose the type of measurement. Enter **d** to find the length (distance) of the line, as follows:
Enter an option [Distance/Radius/Angle/ARea/Volume] <Distance>: **d**
5. Pick the two endpoints of the drawing.
Specify first point: (Pick point 1.)
Specify second point: (Pick point 2.)

Notice that AutoCAD reports more than just the length of the line. Also displayed are the lengths along the x and y axes, as well as the angle of the line to the positive x axis.



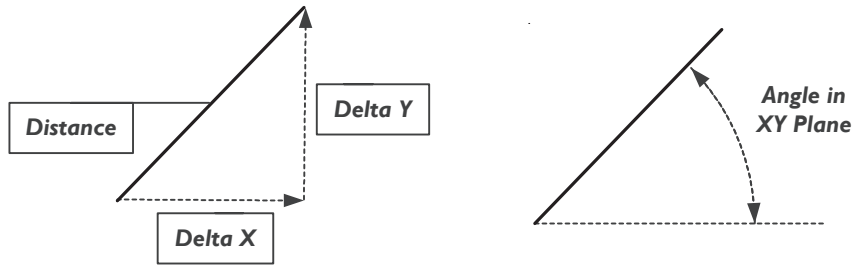
The measurements shown in the drawing area are temporary; as soon as you leave this command, the numbers vanish. For this reason, the information is repeated in the command bar, along with some additional information:

Distance = 9.7188, Angle in XY Plane = 213, Angle from XY Plane = 0

Delta X = -8.1564, Delta Y = -5.2847, Delta Z = 0.0000

(To copy this data for use elsewhere, select the text, and then press **CTRL+C**.)

The x- and y-related information has the following meaning:



Left: The distance, delta X and delta Y reported by the **MEASUREGEOM** command.

Right: The angle in the XY plane.

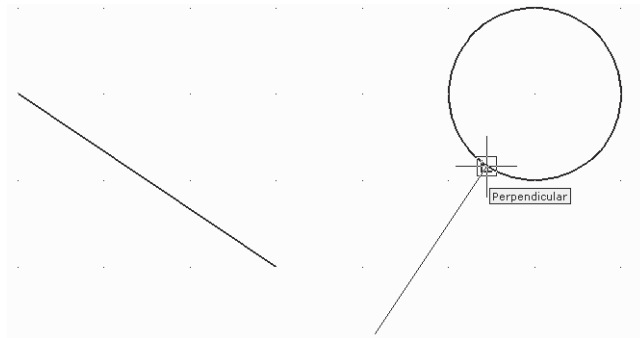
AutoCAD stores the distance in the **DISTANCE** system variable, which you can reuse.

To find the shortest distance between two objects, use the **PERpendicular** object snap, such as between the line and circle illustrated below:

Command: **measuregeom**

Specify first point: **per**

to (Pick a point on the line.)



Specify second point: **per**

to (Pick a point on the circle.)

AutoCAD measures the shortest distance between the two objects, whether or not they would pass by each other. In the figure above, there is a gap between the line and the point of perpendicularity, because AutoCAD “extended” the line segment to find the point of perpendicularity.

When there are more than two objects in the drawing, AutoCAD cannot find the perpendicular until you pick the next point. An ellipsis (...) appears next to the **Perpendicular** icon to indicate the deferment.

Angle Measurement

The **MEASUREGEOM** command's **Angle** option measures angles in a manner different from what you experienced in the tutorial above.

Command: **measuregeom**

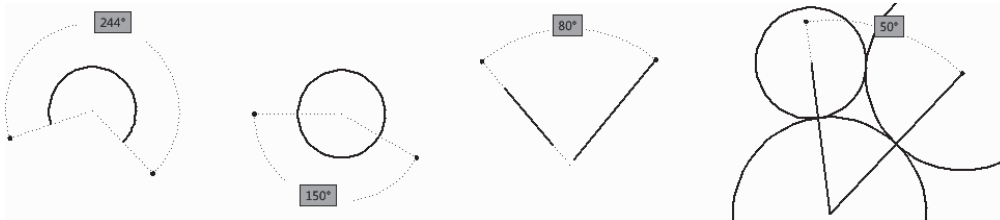
Enter an option [Distance/Radius/Angle/ARea/Volume] <Distance>: **a**

Select arc, circle, line, or <Specify vertex>:

Each of these options is handled differently:

- **Arc** option measures the angle between the two endpoints of the arc.
- **Circle** option measures the angle between two points that you pick on the circle.

- **Line** option measures the angle between two lines.
- **Specify Vertex** option measures the angle between any two points in the drawing, after you pick the vertex point.



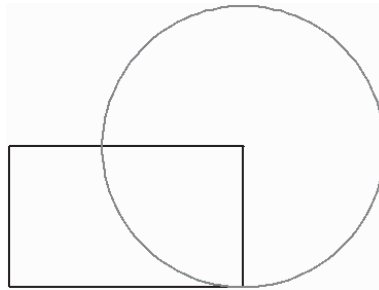
Left to right: Measuring the angles between...

Endpoints of an arc. Two points on a circle. Between two lines. Between the center points of a circle and two arcs.

TUTORIAL: REPORTING AREAS AND PERIMETERS

The **Area** option measures the area and perimeter of closed objects. The list includes circles, ellipses, open and closed splines, open and closed polylines, polygons, regions, hatch patterns, and 3D solids — as well as arbitrary areas of picked points. You can add and subtract many areas, and have AutoCAD report the total.

1. For this tutorial, draw a rectangle that overlaps a circle.



2. To find the area and perimeters of objects, start the **MEASUREGEOM** command, and then enter **ar** for the **Area** option:

Command: **measuregeom**

Enter an option [Distance/Radius/Angle/ARea/Volume] <Distance>: **ar**

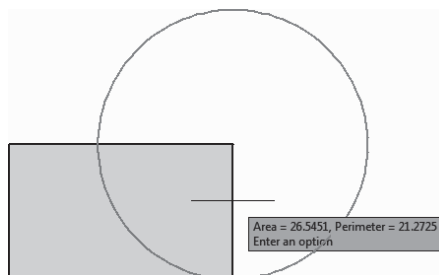
3. In all cases, AutoCAD prompts you to pick objects or points. For this tutorial, enter **o** for the **Objects** option. This allows you to choose any closed object, except self-intersecting ones.

Specify first corner point or [Object/Add area/Subtract area/eXit] <Object>: **o**

4. AutoCAD prompts you to select objects. Choose the rectangle:

Select objects: *(Select the rectangle.)*

Notice that the interior of the rectangle turns green. AutoCAD is confirming that it has measured its area.



5. Press **ESC** to exit the command.

Adding and Subtracting Areas

6. To add areas, repeat the command, and then use the **Area** option's **Add area** option, as follows:

Command: **measuregeom**

Enter an option [Distance/Radius/Angle/ARea/Volume] <Distance>: **ar**

Specify first corner point or [Object/Add area/Subtract area/eXit] <Object>: **a**

7. Specify the **Object** option:

Specify first corner point or [Object/Subtract area/eXit]: **o**

8. Select the rectangle, and then the circle. As you do, notice that AutoCAD reports the individual areas and perimeters and total area in the command bar.

(ADD mode) Select objects: *(Select the rectangle.)*

Area = 26.5451, Perimeter = 21.2725

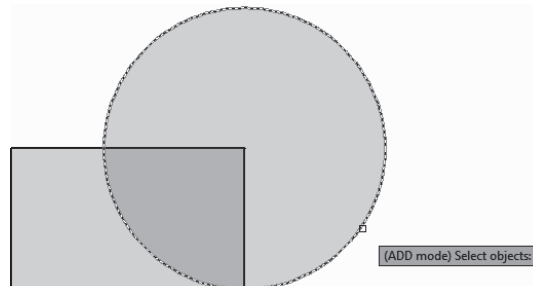
Total area = 26.5451

(ADD mode) Select objects: *(Select the circle.)*

Area = 50.2668, Circumference = 25.1331

Total area = 76.8118

Notice that the overlapping area is colored a darker green.



9. To subtract areas, follow these steps:

- a. Press **ENTER** to exit add mode:

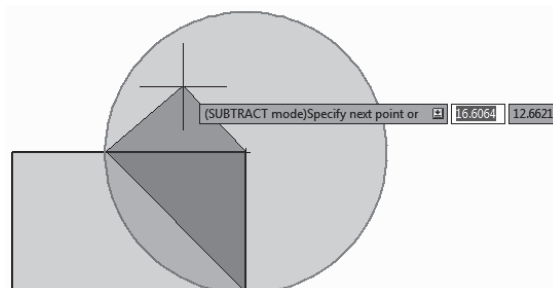
(ADD mode) Select objects: *(Press Enter.)*

- b. Enter **s** to specify **Subtract** area mode:

Specify first corner point or [Object/Subtract area/eXit]: **s**

- c. This time, you'll pick points instead of objects:

Specify first corner point or [Object/Add area/eXit]: *(Pick a point, such as at the intersection of the rectangle and circle.)*



(SUBTRACT mode)Specify next point or [Arc/Length/Undo]: *(Pick another point.)*
(SUBTRACT mode)Specify next point or [Arc/Length/Undo]: *(Pick another point.)*

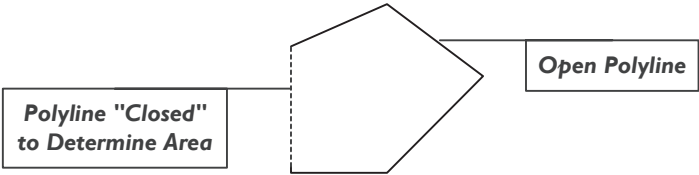
Notice that AutoCAD is coloring the subtraction area in pink.

10. Press **ESC** to end the command.

AutoCAD stores the area and perimeter measurements in the **AREA** and **PERIMETER** system variables. (**AREA** is one of a few system variables with the same name as a related command, and so you must use the **SETVAR** command to access the value stored in the system variable. Entering **AREA** at the ‘Command:’ prompt executes the command.)

Command: **setvar**
Enter variable name or [?]: **area**
AREA = 2.0316 (read only)

You can measure open objects like open polylines, polyarcs, and splines (but not arcs or connected lines). AutoCAD measures the area by temporarily “closing” the open object, as illustrated below.



When it comes to perimeters, however, open objects have shorter perimeters than closed ones of the same area. That’s because AutoCAD measures the length of the open objects without closing its perimeter. The table below lists the data reported by the **MEASUREGEOM**’s Area option:

Objects	Area Measurement
Circles	Area, circumference.
Ellipses, polygons	Area, perimeter.
Regions	Area, perimeter.
Closed polylines and closed splines	Area, perimeter.
Open polylines and open splines	Area, length.
Wide polylines	Area and length/perimeter along centerline.
3D solids	Area of all faces.
Hatch patterns	Area, perimeter.



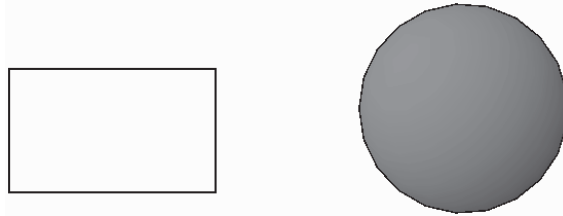
Notes Here’s a quick way to find the area of multiple irregular areas: hatch them with a single pattern, and then look up the area with the Properties palette.

Alternatively, use the **BOUNDARY** command and then find the area from the Properties palette.

TUTORIAL: REPORTING VOLUMES

The **Volume** option measures the volume of 2D and 3D objects. In the case of 2D objects, you are prompted to specify the height. This makes the command useful for measuring room volumes on 2D floorplan, for example: measure the area of the room, and then specify a height of 8' to arrive at the room's volume.

1. For this tutorial, draw a 2D rectangle and a 3D sphere with the **SPHERE** command.



2. The rectangle represents the floor of a room. To find its volume, start the **MEASUREGEOM** command, and then enter **v** for the Volume option:

Command: **measuregeom**

Enter an option [Distance/Radius/Angle/ARea/Volume] <Distance>: **v**

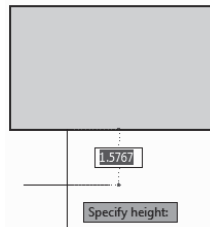
3. Use the **Object** option to select the rectangle, as follows:

Specify first corner point or [Object/Add volume/Subtract volume/eXit] <Object>: **o**

Select objects: *(Choose the rectangle.)*

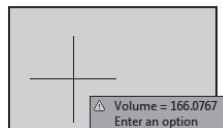
4. Notice that AutoCAD prompts you for the height. Enter **8**.

Enter height: **8**



AutoCAD reports the volume in the tooltip and in the command bar.

Volume = 166.0767



5. Press **ESC** to end the command.

Volumes of 3D Objects

6. To find the volume of 3D objects, repeat the command and its **Volume** option.

Command: **measuregeom**

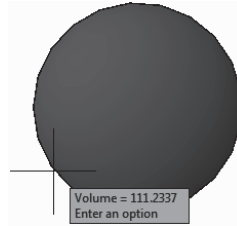
Enter an option [Distance/Radius/Angle/ARea/Volume] <Distance>: **v**

Specify first corner point or [Object/Add volume/Subtract volume/eXit] <Object>: **o**

7. Select the sphere.

Select objects: *(Choose the sphere.)*

8. AutoCAD reports the volume — without asking for a height.

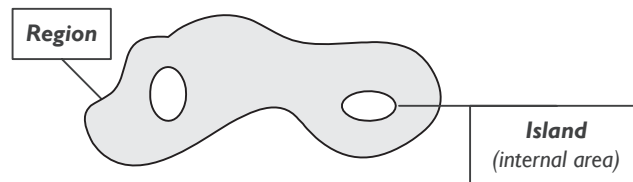


9. Press **ESC** to end the command.

MASSPROP

The **MASSPROP** command reports the area, perimeter, centroid, and so on, of 2D regions and 3D solid models. It does not work with 3D mesh objects or other 2D or 3D objects not made with ShapeManager kernel.

The boundary must be made from a single region object. You can use the **SUBTRACT** command to remove *islands* from regions.



TUTORIAL: REPORTING INFORMATION ABOUT REGIONS

- For this tutorial, draw a closed polyline shaped like the one illustrated above.
- Use the **REGION** command to turn the polyline into a region object.
- To find out information about the region, start the **MASSPROP** command:
 - In the ribbon's Tools tab, choose the **Mass Properties** button in the Inquiry panel.
 - At the 'Command:' prompt, enter the **massprop** command.

Command: **massprop** (Press ENTER.)

- In all cases, AutoCAD prompts you to select objects.

Select objects: (Select region objects.)

Select objects: (Press ENTER to end object selection.)

AutoCAD switches to the Text window, to display a report similar to the following:

```

----- REGIONS -----
Area: 1.7211
Perimeter: 14.3193
Bounding box: X: 1.2616 -- 3.7327
               Y: 0.9440 -- 3.0999
Centroid: X: 2.6010
           Y: 2.2224
Moments of inertia: X: 9.0893
                   Y: 12.3840
Product of inertia: XY: 9.9040
Radii of gyration: X: 2.2981
                  Y: 2.6825
  
```

Principal moments and X-Y directions about centroid:
 I: 0.5772 along [0.9653 -0.2612]
 J: 0.7529 along [0.2612 0.9653]

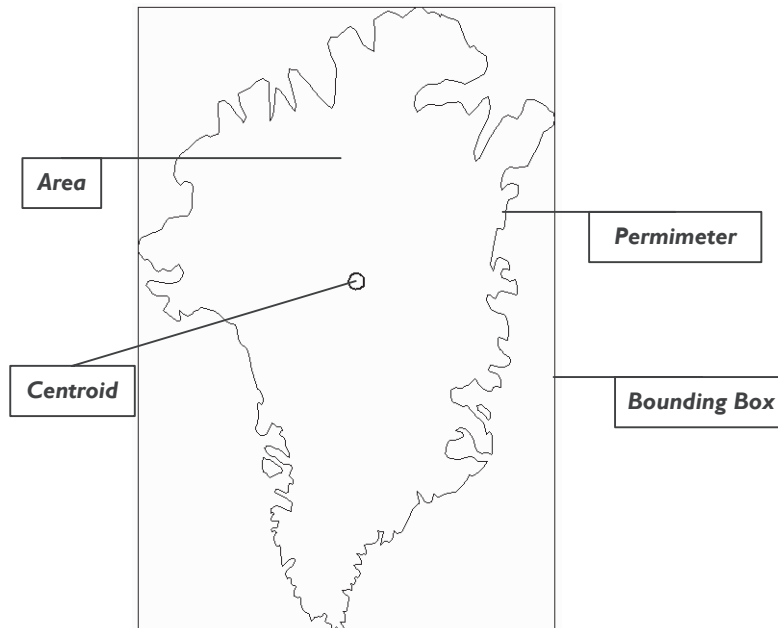
All values are relative to the origin in the current UCS.

5. AutoCAD asks if you wish to save the data to a file.
 Write analysis to a file? [Yes/No] <N>: **y**
6. If you answer **Y**, AutoCAD displays the Create Mass and Area Properties File dialog box. Enter a file name, and then click **Save**. AutoCAD creates the *.mpr* mass properties report file.
7. Press **F2** to return to the drawing window.

The same command works with 3D solid models, and provides a similar report.

Finding the Center

The centroid value is useful for finding the geographic center of irregularly shaped objects, like maps, such as that of Greenland illustrated below. The bounding box is the smallest rectangle that completely surrounds the region. The area and perimeter values are the same as those reported by the AREA, LIST, and PROPERTIES commands.





LIST

The **LIST** command reports information about selected objects in the command bar.

TUTORIAL: REPORTING INFORMATION ABOUT OBJECTS

- To find out information about objects, start the **LIST** command:
 - In the ribbon's Home tab, choose the **List** button in the Properties panel.
 - At the 'Command:' prompt, enter the **list** command:

Command: **list** (Press ENTER.)
 - Alternatively, enter the **ls** alias at the 'Command:' prompt.
- In all cases, AutoCAD prompts you to select objects. While you can select more than one object, it is better to pick just one, because of the amount of information generated.

Select objects: (Select one object.)

Select objects: (Press ENTER to end object selection.)

AutoCAD reports on the object(s) in the Text window.

```
CIRCLE Layer: "0"
Space: Model space
Handle = 15f22
center point, X=1231.9752 Y=-1589.5176 Z= 0.0000
radius 593.9697
circumference 3732.0214
area 1108353.7564
```

- You can select the text, and then use **CTRL+C** to copy it to the Clipboard for pasting into other documents.

Press **F2** to return to the drawing window.

AutoCAD reports information that is specific to each object. For instance, circles return the following information:

Line #	Example	Information Reported
1	CIRCLE	Type of object.
	"0"	Layer on which the object resides.
2	Model space	Space in which the object resides: model or paper space.
3	9D	Handle (unique hexadecimal identifier assigned by AutoCAD.)
4	0.5,7.5,0.0	X, y, z coordinates of the object's location.
Other		Additional geometric information specific to the object: Area, perimeter, length. Color, linetype, and lineweight, if not set to BYLAYER. Thickness, if not zero.

The geometric information varies among objects, as illustrated by the table below. For complex objects, the listing can get very long, because AutoCAD reports on every subpart.

Object	Information Reported
Arc	Center point, Radius, Start angle, End angle, Length.
Box	Position, Length, Width, Height, Rotation.
Circle	Center point, Radius.
Ellipse	Center, Major Axis, Minor Axis, Radius Ratio.
Hatch	Pattern name, Scale, Angle, Associativity, Area, Origin.
Helix	Position, Length, Base radius, Top radius, Turns, Step Dist, and Axis.
Line	From point, To point, Length, Angle in XY Plane, Delta X, Delta Y, Delta Z.
Multileader	Leader Type, Content Type, Landing, Leader Number, Vertex, Annotative scale.
PlaneSurface	U-isolines, V-isolines, Bounding Box.
Point	Point.
Polyline	Constant width (or Starting width and Ending width), Area, Perimeter.
Polyarc	Bulge, Center, Radius, Start angle, End angle.
Sphere	Position, Radius.
Spline	Area, Circumference, Order, Properties, Parametric Range, Number of control points, Control Points, Number of fit points, User Data, Start Tangent, End Tangent.
Text	Style, Annotative, Font file, Start point, Height, Text, Rotation angle, Width scale factor, Obliquing angle, Generation
etc.	



Note The **LIST** command is more useful than **AREA** for individual objects, because it provides more information. For example, **AREA** reports the area and circumference of circles, but **LIST** also reports their radius.

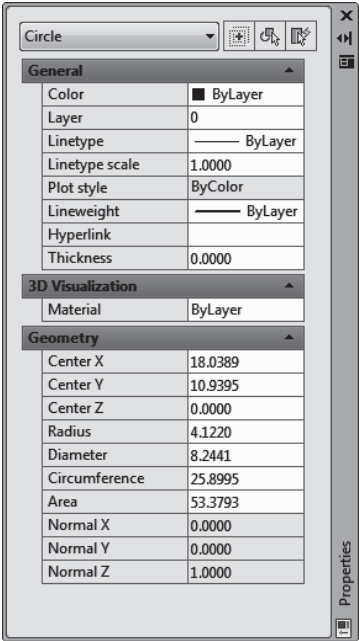


PROPERTIES

The **PROPERTIES** command reports on objects in a format easier to digest and more complete than **LIST**. Compare the data for the circle shown at right with that listed on the previous page.

When you select all objects in the drawing, the Property window reports the total number.

A drawback is that the data listed by the Properties palette cannot be copied to the Clipboard.

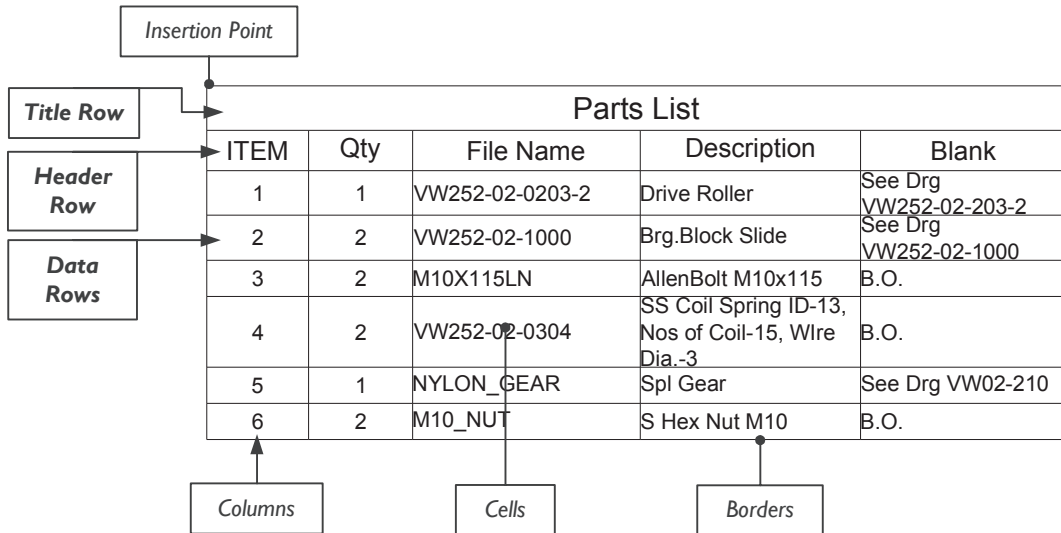


TABLE

The **TABLE** command creates the outline structure of tables, following which you fill cells with text, formulas, and drawings.

Tables consist of *rows* and *columns*. AutoCAD descriptively names three kinds of rows, names that are used when creating table styles: title, header, and data row. Columns don't get special names.

The topmost row is usually reserved for the title of the table, such as "Parts List" illustrated below. AutoCAD calls this the "title row."



The second row typically labels the columns, such as "ITEM" and "Qty" (short for "quantity"). AutoCAD calls this the "header row."

All other rows in the table are called "data rows."

The rows and columns of tables define *cells*, just as in spreadsheets. Each cell contains one piece of information, either some text (or numbers or formulas) or an image of a block or drawing.

Tables are not static; you can change them at any time. You can add and remove rows and columns, merge cells to make them longer or taller, format cells to add color or more spacing, and export the table data for use by other software. Tables can receive data, from attributes in blocks or external files. The title, header, and data rows can be formatted differently from each other, as can individual cells. Tables are changed through grips editing, the Properties palette, and commands found on shortcut menus.



Tables have insertion points, just like text and blocks. The *insertion point* is the anchor from which

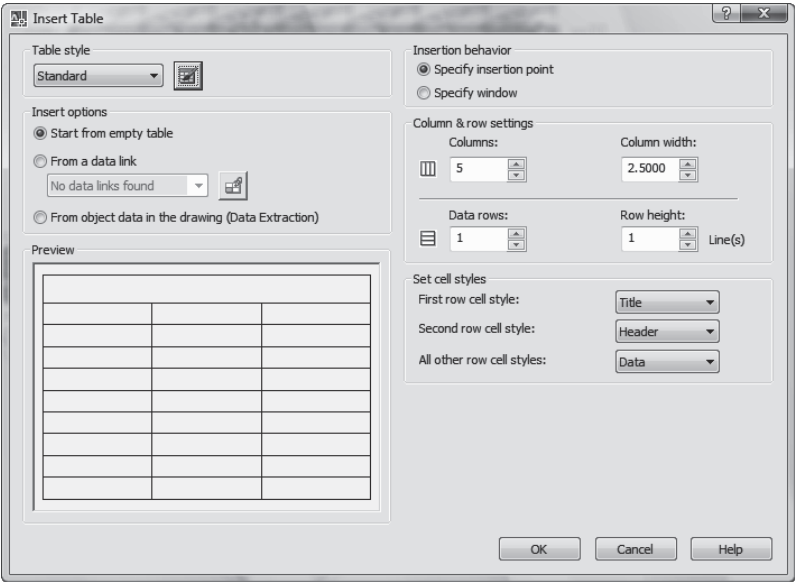
the table grows and shrinks. The insertion point is typically located at the table’s upper-left corner. When you add rows, the table grows downward from its insertion point.

In some drawings, it makes more sense to have the insertion point at the bottom corner, so that the table grows upwards. Below, the title and header rows are at the bottom of the table.

BASIC TUTORIAL: PLACING TABLES

- 1. To place tables in drawings, start the **TABLE** command with one of these methods:
 - In the ribbon’s Home tab, choose the **Table** button in the Annotation panel.
 - At the ‘Command:’ prompt, enter the **table** command.
Command: **table** (Press ENTER.)
 - Alternatively, enter the **tb** alias at the ‘Command:’ prompt.

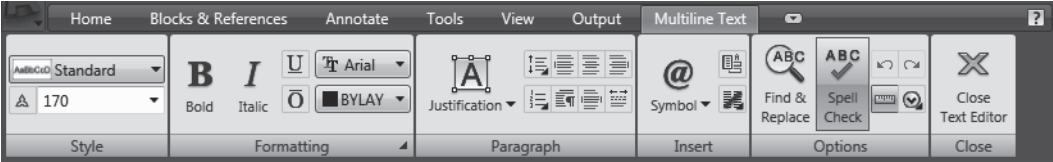
In all cases, AutoCAD displays the Insert Table dialog box.



- 2. Ignore all the options, and click **OK**.
- 3. AutoCAD prompts you for the insertion point:
Specify insertion point: (Pick a point.)

AutoCAD displays either the Multiline Text tab on the ribbon or Text Formatting bar (identical to that displayed by the **MTEXT** command). The **TABLETOOLBAR** system variable determines which interface appears:

When **TABLETOOLBAR** = 2 (default), the Multiline Text tab appears on the ribbon:



When **TABLETOOLBAR** = 1, the Text Formatting bar appears in the drawing area:



In either case, the title row is highlighted. This means that when you start typing, text is entered in that cell.

	A	B	C	D	E
1	Table of Parts				
2					
3					

4. Enter a title for the table:

Table of Parts

5. Press the **TAB** key. Notice that AutoCAD highlights the next cell.

6. Enter a header name for the column:

Item Number

	A	B	C	D	E
1	Table of Parts				
2	Item Number				
3					

7. Press the **TAB** key, and the highlight jumps to the next cell. (Press **SHIFT+TAB** to move to the previous cell.) You can also use the cursor keys to move to other cells.

Enter the next header name:

Quantity (press **TAB**.)

8. AutoCAD auto-fill cells when you drag the cursor across cells in a specific manner. Here's how:

- a. Enter "1" (one) in the cell under Item Number.
 b. Select the cell containing the number "1." (Use a single click.) Notice the cell has five grips:

Square grips resize the cell.

Diamond grip performs auto-fill.

	A	B	C	D	E
1	Table of Parts				
2	Item Number	Quantity			
3					

- c. Drag the diamond grip across blank cells. Notice that the tooltip displays the number that will be auto-filled. Numbers are incremented; if you auto-fill with a cell containing a word, the same word is repeated in each cell.

	A	B	C	D	E
1	Table of Parts				
2	Item Number	Quantity			
3					

- d. Release the mouse button. Notice that the numbers 2 - 5 fill the cells.

	A	B	C	D	E
1	Table of Parts				
2	Item Number	Quantity			
3	1	2	3	4	5

9. When done filling the table's cells, click outside of the table to exit the command.



Notes When you press **TAB** in the last cell, AutoCAD automatically creates another row.

To move around the table, you can also press the four cursor keys on the keypad. Pressing **ENTER** moves the highlight down the table.

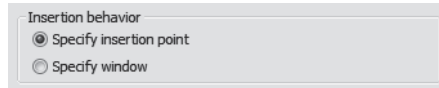
To create line breaks (where text is forced to start on the line below), press **ALT+ENTER**. To change the text style of a cell quickly, select a text style from the **Text Style** droplist on the Standard toolbar.

ADVANCED TUTORIALS: INSERTING TABLES

The Insert Table dialog box contains several options for controlling the initial size of the table.

Insertion Behavior

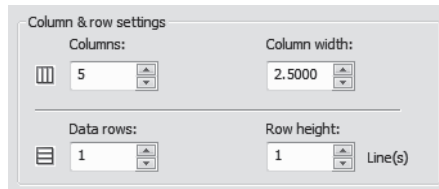
The Insertion Behavior section of the dialog box provides two basic methods of specifying the overall size of the table:



- **Specify Insertion Point** — you specify where the table must start; AutoCAD determines the size of the table.
- **Specify Window** — you specify the area into which the table must fit; AutoCAD determines how to size the cells to fit.

Column & Row Settings

The numbers you specify for the table's rows and column are not critical, because you can always add and remove rows and columns later, as well as change the width and height of cells. Still, the Column & Row Settings section provides two ways of changing the size of the cells that make up the rows and columns:



- **Columns** and **Data Rows** — you specify the number of columns and rows; AutoCAD determines the size based on column width and row height.
- **Column Width** and **Row Height** — you specify the width of columns and height of rows; AutoCAD determines the number of rows and columns to create.



Note The **Row Height** is measured in “lines.” A *line* is defined as the text height + top margin + bottom margin. The default value is 0.3 units, which comes from 0.18 (text height) + 0.06 (top margin) + 0.6 (bottom margin).

Table and Cell Styles

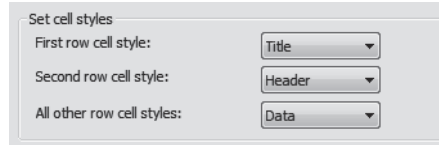
When the drawing already has table styles defined, you can select one from the **Table Style Name** droplist. (You can use DesignCenter to borrow table styles from other drawings.)



If you want to create or modify a table style, click the  button, which displays the Table Style dialog box.

Set Cell Styles

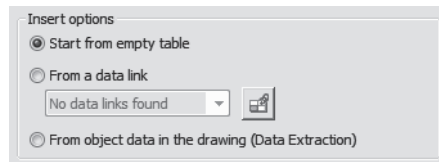
You can specify style override with the Set Cell Styles options. Although called “cell styles,” the styles apply to entire rows: the first row uses the “Title” style by default, the second row the “Header” style, and remaining rows the “Data” style.



These styles can be modified with the **TABLESTYLE** command, as detailed later in this chapter.

Data Source

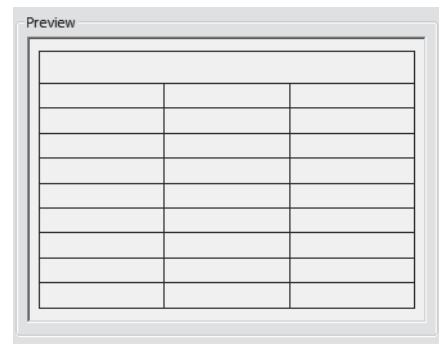
Tables can be created from data in the drawing or from external data files, such as spreadsheets. This is an advanced topic not covered by this book.



Tables can be inserted from spreadsheet programs: copy the data (using **CTRL+C**), and then paste it (**CTRL+V**) into the AutoCAD drawing. When spreadsheets are copied from Excel XP, they are pasted as table objects. For Excel, use the **PASTESPEC** command, and then select “AutoCAD Entities”; the spreadsheet data are pasted as a table. For other spreadsheets, the data are pasted as mtext.

Preview

The preview image, unfortunately, does not reflect the number of rows, columns, and sizes specified; it reflects only the visual characteristics of the current table style, such as fonts and colors.



EDITING TABLES

AutoCAD provides many options for editing tables. The methods of editing tables and cells include:

- **TABLEEDIT** command edits the text of the selected cell.
- **TABLETOOLBAR** system variable toggles display of the table editing toolbar.
- **TINSERT** command inserts blocks and drawings in cells.
- **PROPERTIES** command changes many properties of tables.
- **TABLEEXPORT** command exports the table data as a comma-delimited text file.

I find it curious that a command named **TABLEEDIT** edits *cells* (and then just the text in cells) and

not the table itself. Another curiosity: the spelling of the **TABLEDIT** command has one “e,” while **TABLEEXPORT** has two.

Many other table and cell editing and property commands are “hidden” in shortcut menus. For the following tutorials, open the *table.dwg* file from the Companion CD, the drawing of a table.

TableEdit Command

The **TABLEDIT** command (one “e”) edits the text of selected cells. It edits a cell at a time.

- 1. To edit text in a table cell, start the **TABLEDIT** command with one of these methods:
 - At the ‘Command:’ prompt, enter the **tabledit** command. AutoCAD prompts you to select a cell:
Command: **tabledit** (Press ENTER.)
Pick a table cell: (Select a single cell.)
 - Double-click the cell you wish to edit; this avoids the need to enter the command.
 - Alternatively, select a cell, and then click the right mouse button. From the shortcut menu, select **Edit Text**.

In all cases, AutoCAD highlights the selected cell in gray:

6	10	2	VW252-02-1102	Upper Bush	See Drg#VW252-02-1100_U
7	12	1	VW252-02-0201-N	M.S. Pipe	See Drg#VW252-02-0203-2
8	13	1	VW252-02-0202	Upper End SS Shaft	See Drg#VW252-02-0203-2
9	14	1	VW252-02-0203N	Rubber Coating	See Drg#VW252-02-0203-2
10	15	1	VW252-02-0204	Lower End SS Shaft	See Drg#VW252-02-0203-2

- 2. Make your editing changes; see the **MTEXT** command in the earlier chapter. You can edit the text, or change properties, for example, make the text boldface, insert a symbol, or change its justification. (These are called “local overrides.”)
- 3. Use the **TAB** key or the cursor keys to edit other cells.
- 4. When you are finished editing the text, press the **ESC** key, or click outside of the table.

TableToolbar System Variable

The **TABLETOOLBAR** system variable controls the display of the Table toolbar used for editing tables. (The Table tab is always displayed when the ribbon is open, no matter the setting of this system variable.)

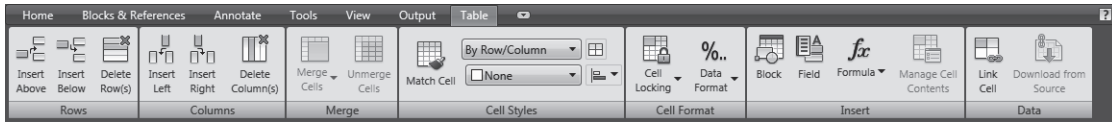
Command: **tabletoolbar** (Press ENTER.)
Enter new value for TABLETOOLBAR <2>: (Type **2** or **1** or **0**.)

- 0** — Table toolbar is never displayed, even when ribbon is closed.
- 1** — Table toolbar is always displayed, and Table tab is displayed when ribbon is open.
- 2** — Table toolbar is displayed only when ribbon is closed; Table tab is only displayed when the ribbon is open (default).

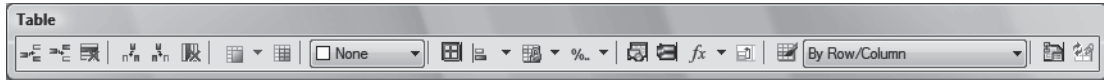
The values of the system variable can be somewhat confusing, so this table summarizes them:

TABLETOOLBAR	Toolbar	Ribbon
0	Not displayed.	Displayed when open.
1	Displayed.	Displayed when open.
2	Displayed when ribbon is closed.	Displayed when open.

The Table tab appears when you select one or more cells, and looks like this:



The Table toolbar looks like this:



The options are very similar to the right-click menu described later in this chapter.

TInsert Command

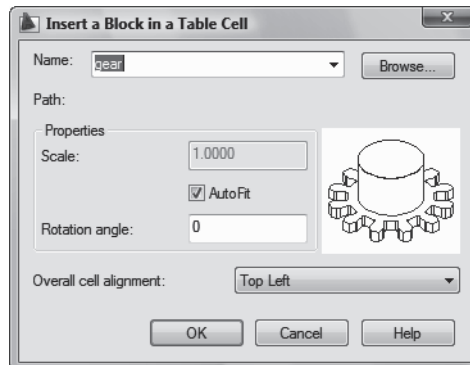
The TINSERT command inserts blocks and drawings in cells (short for “table insert”). Cells can contain blocks or drawings together with text.

1. To insert blocks into cells, start the **TINSERT** command with one of these methods:
 - At the ‘Command:’ prompt, enter the **tinsert** command. AutoCAD prompts you to select a cell:

Command: **tinsert** (Press ENTER.)

Pick a table cell: (Select a single cell.)
 - Alternatively, select a cell, and then click the right mouse button. From the shortcut menu, select **Insert Block**.

In both cases, AutoCAD displays the Insert a Block in a Table Cell dialog box.



2. Change the dialog box settings, as follows:

Name	gear
AutoFit	(turned on)
Rotation Angle	0
Overall Cell Alignment	Middle Center

“Gear” is the name of a block definition provided in the *table.dwg* file.

Browse lets you insert an entire drawing, including .dwg and .dxf files.

Overall Cell Alignment is like text justification; the **Middle Center** cell alignment option centers the block in the cell; you can choose any other alignment.

AutoFit determines the size of the block:

- ☒ Block scaled automatically to fit the cell.
- ☐ Cell resized to fit the block; **Scale Factor** option becomes available.

Rotation angle rotates the block; 90 degrees, for example, turns the block on its left side.

3. Click **OK**.

AutoCAD places the block next to the text in the cell.

Parts List		
Number	Description	Comments
	Spl. Gear	See Drg#/W202-210
gear	Helical Gear	
101	Upper Plate	See Drg#/W252-02-1100_U
102	Upper Bush	See Drg#/W252-02-1100_U



Note To make it appear as if a single cell bridges two cells (yet remaining two cells), change the border property so that the line between two cells is invisible.

	Description	Comments
	Spl. Gear	See Drg#/W202-210
	Helical Gear	
	Upper Plate	See Drg#/W252-02-1100_U
	Upper Bush	See Drg#/W252-02-1100_U

Properties Command

The PROPERTIES command changes many (but not all) properties of tables and cells.

Table

General

Color

ByLayer

Layer

0

Linetype

ByLayer

Linetype scale

1.0000

Plot style

ByColor

Lineweight

ByLayer

Hyperlink

3D Visualization

Material

ByLayer

Table

Table style

SHEET INDEX

Rows

14

Columns

5

Direction

Down

Table width

129.8921

Table height

79.4436

Geometry

Position X

12.1554

Position Y

81.9791

Position Z

0.0000

Table Breaks

Enabled

No

Direction

Right

Repeat top labels

No

Repeat bottom labels

No

Manual positions

No

Manual heights

No

Break height

0.0000

Spacing

0.5156

Table

Cell

Cell style

By Row/Column

Row style

Data

Column style

(none)

Cell width

28.8819

Cell height

6.1200

Alignment

Middle Left

Background fill

None

Border color

ByBlock

Border lineweight

ByBlock

Border linetype

ByLayer

Horizontal cell margin

0.0600

Vertical cell margin

0.0600

Cell locking

Unlocked

Cell data link

Not linked

Content

Cell type

Text

Contents

Rubber Coating

Text style

Arial

Text height

2.0000

Text rotation

0

Text color

ByBlock

Data type

Text

Format

(none)

Precision

Additional format

Left: Properties for tables.
Right: Properties for cells.

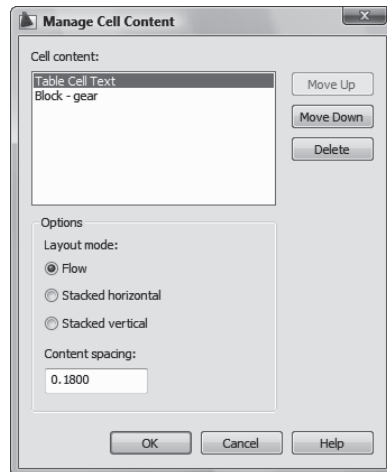
The Properties palette has different displays, depending on what part of the table you select.

Grips Editing

Grips editing changes the size of the table, rows, and columns — depending on where you click, and where you grip. Once the cell(s) or table is selected, blue grips appear. You can use the grips to change the size of the table, the rows, and the columns — again, depending on which grips you drag.

Managing Cell Content

In the past, AutoCAD restricted cells to one piece of content, either text or a block. As of AutoCAD 2008, cells can contain different types of content. A new dialog box controls the display order.



(There appears to be no command for accessing this dialog box. When a cell contains two or more objects, right-click, and then choose **Manage Content** from the shortcut menu.)

The Cell Content dialog box lists the contents of the selected cell. Use the **Move Up** and **Move Down** buttons to change the display order. Click **Delete** to erase the selected item.

Flow — determines where to place the cell contents according to the width of the cell; some might be next to each other; others might be stacked on top of each other.

Stacked Horizontal — lays out cell content horizontally.

Stacked Vertical — stacks cell content vertically.

Content Spacing — specifies the width between cell content items.

TableExport Command

The TABLEEXPORT command (two “e”s) exports the table data as CSV-format files (short for “comma-separated values”).

This means the table data are saved as an ASCII text file, with data separated by commas, one line of data per table row. Here is an example:

```
"VW252 - Washing Unit Drawing Sheets"
"Sheet Number","Sheet Title"
"00","Cover Sheet"
"01","VW252-Washing-Unit"
"03","Drive Roller Sub Assy"
```

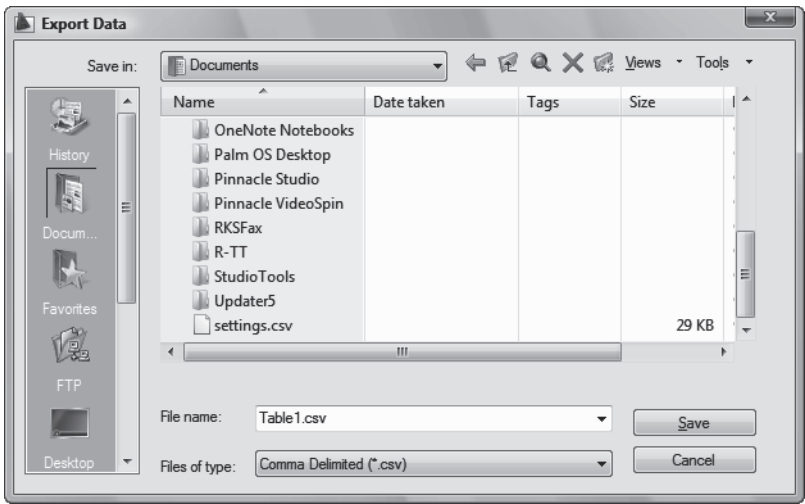
The .csv files can be imported into just about any spreadsheet and database program, as well as some word processors and other software.

AutoCAD exports the table data literally. If the table's direction is up, you may want to use the **PROPERTIES** command to change the table's **Direction** property to **Down** before exporting the table — otherwise the title and header rows appear at the end of the .csv file for tables with an Up direction.

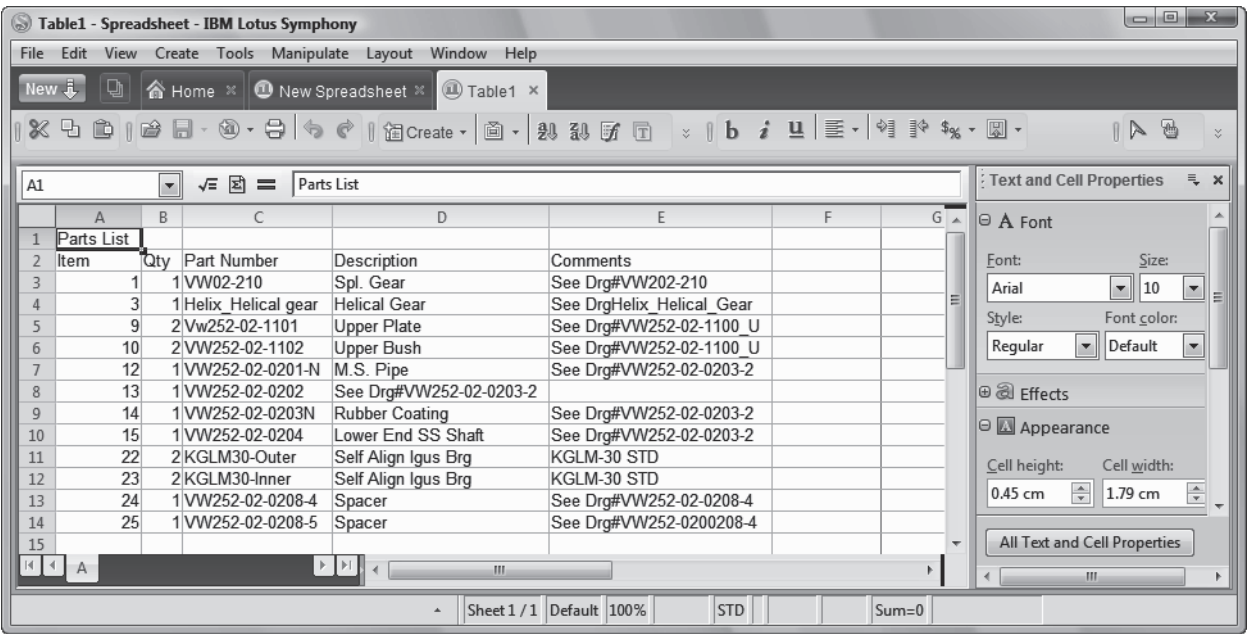
- 1. To export tables from drawings, start the **TABLEEXPORT** command with one of these methods:
 - At the 'Command:' prompt, enter the **TABLEEXPORT** command. AutoCAD prompts you to select a table:

Command: **tableexport** (Press ENTER.)
Select a table: (Select a single table.)
 - Alternatively, select a table, and then click the right mouse button. From the shortcut menu, select **Export**.

In either case, AutoCAD displays the Export Data dialog box.



- 2. Specify the path, enter a file name, and then click **OK**.
AutoCAD exports the data. You can open the file in a spreadsheet or database program.



TABLESTYLE

The **TABLESTYLE** command defines named table styles, allowing you to change the look of tables quickly, for example giving them thicker lines, color, and other fonts.

Tables have styles similar to text and dimensions. I find the default style included with AutoCAD, called “Standard,” quite ugly; you’ll want to change it right away with this command — or use DesignCenter to borrow table styles from other drawings.

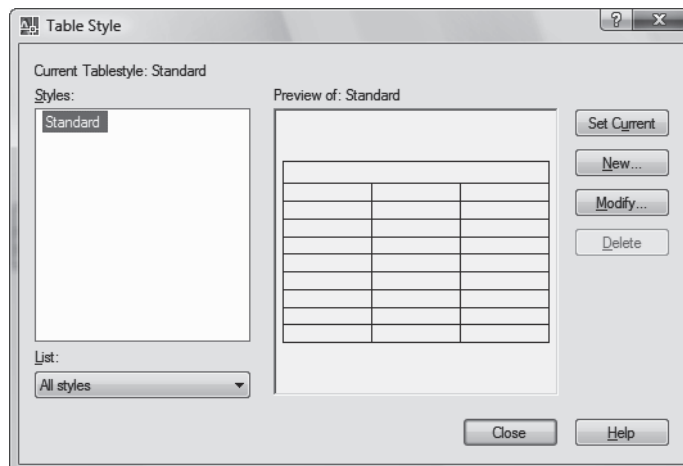
As of AutoCAD 2008, table styles include *cell styles*, which define sub-styles for individual cells. AutoCAD includes predefined cell styles named Data, Header, and Title. You can edit these cell styles, as well as create new ones.

This command allows you to change the properties of cells, header rows, and title rows independently of each other. In addition, you can specify the color and thickness of lines, color of text and color behind cells, margin widths, justification within cells, and more.

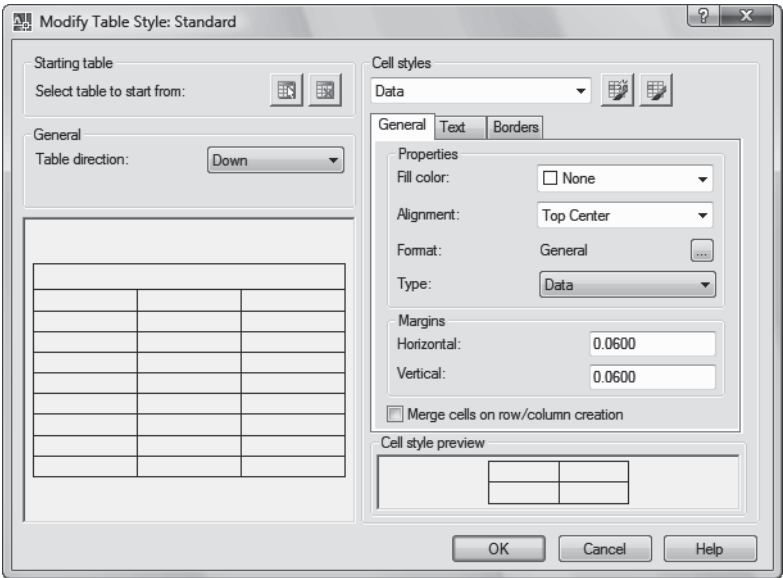
BASIC TUTORIAL: CREATING TABLE STYLES

1. From the Companion CD, open the *tablestyle.dwg* file, a drawing of a table with the Standard style.
2. To modify the table style, start the **TABLESTYLE** command with one of these methods:
 - In the ribbon’s Home tab, choose the **Table Style** button in the Annotation panel.
 - At the ‘Command:’ prompt, enter the **tablestyle** command.
Command: **tablestyle** (Press ENTER.)
 - Alternatively, enter the **ts** alias at the ‘Command:’ prompt.

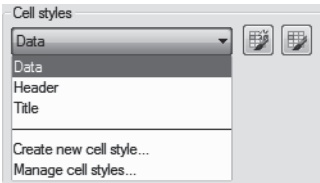
In all cases, AutoCAD displays the Table Style dialog box.



3. To improve the look of the Standard style, click **Modify**. AutoCAD displays the Modify Table Style dialog box.

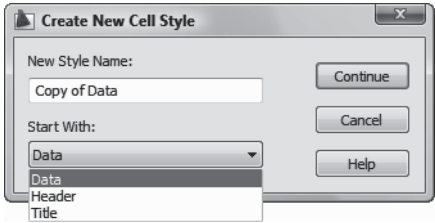


The most important item in this dialog box is the **Cell Styles** droplist, because it displays the names of cell styles — the styles that affect the looks of cells.



- Data** — generic style for data cells, all cells that aren't headers or titles.
- Header** — cell style for header cells, usually in the table's second row.
- Title** — cell style for title cells, usually the table's first row.

Create New Cell Style  — displays a dialog box for naming new cell styles. Note that new styles are based on existing ones. Clicking **Continue** returns you to the Table Style dialog box.



7. Click **OK**, and then **Close** to exit the dialog boxes.

You have modified the Standard table style. To see what it looks like, use the **TABLE** command to create a table, and then fill its cells with text.



Note Changing a table style definition retroactively changes existing tables made with the same style.



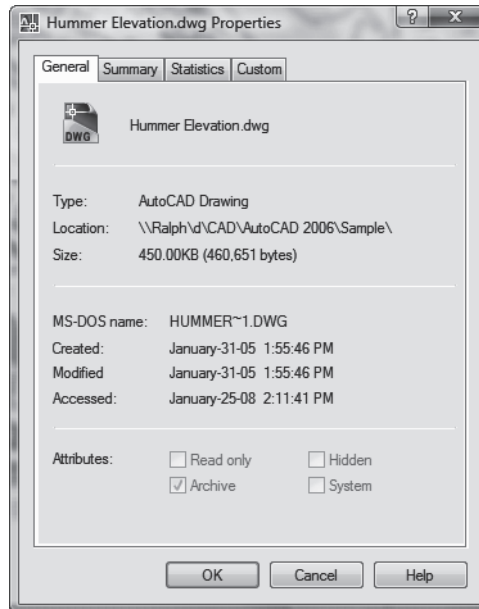
DWGPROPS

The DWGPROPS command reports and stores information about drawing files.

This command displays a dialog box of data related to the file, such as its size and attributes. As well, you can add information about the drawing, such as the name of the drafter and general comments. This data can be searched with AutoCAD's Design Center (ADCCENTER) command and the Windows Explorer's Find (or Search) options.

TUTORIAL: REPORTING INFORMATION ABOUT DRAWINGS

1. To find out information about drawings, start the **DWGPROPS** command:
 - At the 'Command:' prompt, enter the **dwgprops** command.
Command: **dwgprops** (Press ENTER.)
2. AutoCAD displays the Properties dialog box — which is different from the Properties palette (illustrated above). The dialog box is for drawing properties; the palette for object properties.



The General tab shows information about the drawing file's name, location, size, and dates. (If the drawing has not yet been saved, most fields are blank.) Attributes can be changed only by the Windows Explorer's **Property** option.

3. Click the **Summary** tab.

The screenshot shows the 'Summary' tab of a drawing application. It contains the following fields:

- Title: [Text input field]
- Subject: [Text input field]
- Author: [Text input field]
- Keywords: [Text input field]
- Comments: [Large text area with scrollbars]
- Hyperlink base: [Text input field]

You can add data to the fields, such as your name, the subject of the drawing, and general comments.

- **Title** can be different from the drawing's file name, such as the project name.
- **Keywords** are useful when drawings are searched by database programs.
- **Hyperlink Base** is the base URL address used by relative hyperlinks in the drawing.

4. Click the **Statistics** tab.

- **Created** is the date and time the drawing was first created with the **NEW** command; the date and time are retrieved from the **TDCREATE** system variable.
- **Last Modified** provides the date and time the drawing was last edited or had objects added, retrieved from the **TDUPDATE** system variable.
- **Last Saved By** provides the user's login name, retrieved from **LOGINNAME** system variable, which in turn is obtained from Windows.
- **Revision Number** provides the AutoCAD software revision number.
- **Total Editing Time** provides the time the drawing has been open. The name is misleading, because it includes time when the drawing is displayed by AutoCAD without being edited. The value is retrieved from the **TDINDWG** system variable.

The screenshot shows the 'Statistics' tab of a drawing application. It displays the following statistics:

Created:	June-11-96 3:57:20 PM
Modified:	January-31-05 1:55:41 PM
Last saved by:	Autodesk
Revision number:	
Total editing time:	2262 hours, 57 minutes



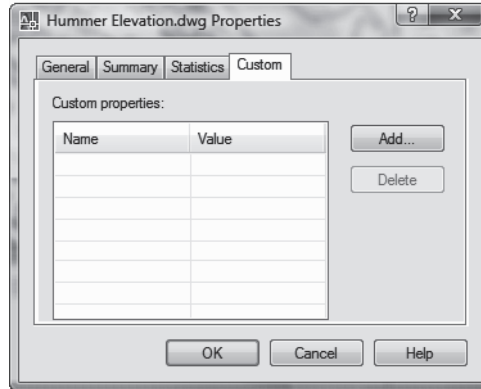
Note If the drawing was last saved by a non-Autodesk software product, an additional message appears in this tab:

This file may have been last saved by a program other than Autodesk software.

In addition, the value of 0 is stored in **DWGCHECK** system variable.

5. Click the **Custom** tab.

You can enter text in up to ten fields, along with a value for each. The custom fields are searchable by the **Find** option of DesignCenter.



To add a field, click **Add**; enter a name for the field and optionally a value. Click **OK**.

To delete a field, select it, and then click the **Delete** button.

6. Click **OK** to close the dialog box.

'SETVAR

The **SETVAR** command reports the settings of system variables (short for “set variables”).

System variables store the state of the drawing and control numerous commands. You can change the value of most system variables, thus changing the state of the drawing (snap, grid, and so on) and the actions of commands (mirrored text, current layer, and so on). Some system variables cannot be changed; they are called “read-only.”

This command can be used transparently during other commands.

You have used this command many times in this and previous chapters, without realizing it. (*History:* Previously, system variables could only be changed with the **SETVAR** command. But then Autodesk made it possible to enter system variables directly at the ‘Command:’ prompt. The only time you must use this command is to examine the list of system variables names and values, or when the name of the system variable is the same as the command, such as **AREA**.)

All of AutoCAD’s system variables are listed in Appendix C of this book.

TUTORIAL: LISTING SYSTEM VARIABLES

1. To list the values of system variables, start the **SETVAR** command:
 - At the ‘Command:’ prompt, enter the **setvar** command:
 Command: **setvar** (Press ENTER.)
 - Alternatively, enter the **set** alias at the ‘Command:’ prompt.
2. In all cases, AutoCAD prompts you to enter the name of a system variable.
 Enter the ? option:
 Enter variable name or [?] <AREA>: ?
3. AutoCAD asks which variables you wish to see. Use wildcard characters:
 - * — means all variables.
 - g*** — means all variables starting with the letter “g.”
 - grid*** — means all variables starting with the word “grid.”

? — means a single character.

Enter variable(s) to list <*>: (Press **ENTER** to list all variables.)

AutoCAD flips to the Text window, and displays the value of all system variables.

```

3DDWFPREC      2
ACADLSPASDOC  0
ACADPREFIX     "C:\Documents and Settings\Administrator\ApplicationData\Aut..." (read only)
ACADVER        "17.0s (LMS Tech)" (read only)
ACISOUTVER     70
AFLAGS         16
ANGBASE        0
ANGDIR         0
APBOX          0
APERTURE       10
AREA           1365453.6670 (read only)
ATTDIA         0
ATTMODE        1
ATTREQ         1
AUDITCTL       0
AUNITS         0
AUPREC         0
AUTOSNAP       39
et cetera...

```

4. With almost 600 system variables, the listing is very long. Press **ESC** to end it early. To return to the drawing window, press **F2**.

'TIME

The **TIME** command reports the time spent editing drawings, and provides timer functions. This command can be used transparently in the middle of other commands.

TUTORIAL: REPORTING ON TIME

1. To find out time-related information about drawings, start the **TIME** command:
 - At the 'Command:' prompt, enter the **time** command.

Command: **time** (Press **ENTER**.)

AutoCAD reports time-related data similar to the following:

Command: **time**

Current time: Wednesday, June 25, 2009 at 2:53:15:890 PM

Times for this drawing:

Created: Wednesday, June 25, 2009 at 1:04:52:812 AM

Last updated: Wednesday, June 25, 2009 at 1:04:52:812 AM

Total editing time: 0 days 13:48:23.203

Elapsed timer (on): 0 days 13:48:23.047

Next automatic save in: <no modifications yet>

2. Press **ENTER** to exit the command:

Enter option [Display/ON/OFF/Reset]: (Press **ENTER**.)

The accuracy of the time depends on your computer's clock. The meaning of the data reported by this command is as follows:

Time	Meaning
Current Time	Current date and 24-hour time to the nearest millisecond.
Created	Date and time drawing was created.
Last Updated	Latest use of the SAVE and QSAVE commands.
Total Editing Time	Cumulative time drawing has been open in AutoCAD, excluding plotting time, and sessions when drawing was exited without saving.
Elapsed Timer	Stopwatch-like timer.
Next Automatic Save In	Minutes remaining until the next automatic save.

REPORTING ON TIME: ADDITIONAL METHODS

The TIME command includes options to redisplay its output and act as a stopwatch.

- **Display** redisplay the time report.
- **ON** starts the timer.
- **OFF** turns off the timer.
- **Reset** resets the timer.

Let’s look at each.

Display

The **Display** option redisplay the time report, with updated times.

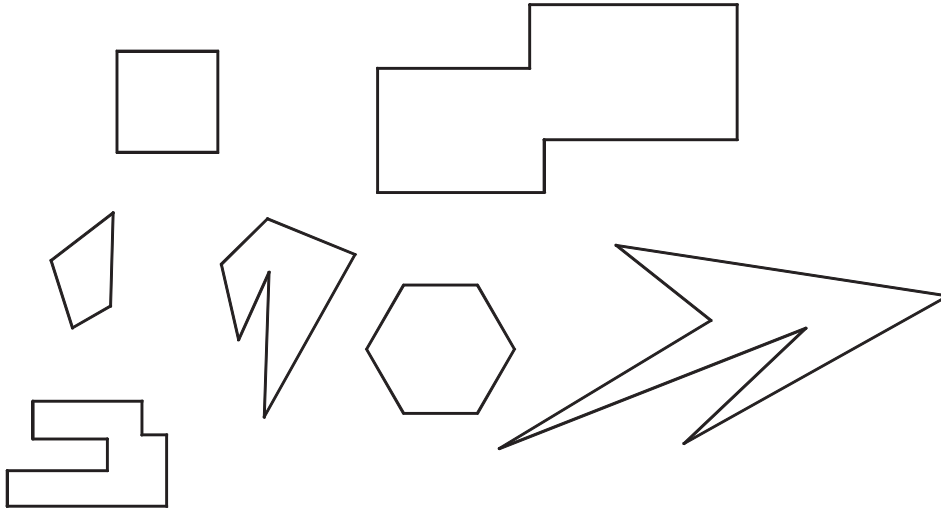
ON, OFF, and Reset

The **ON** option starts the timer. The **OFF** option stops the time, and the **Reset** option sets the timer back to 0 days and 0 time (0 hours, 0 minutes, and 0 seconds).

EXERCISES

1. Start AutoCAD with a new drawing. Use this drawing for all the exercises in this chapter.
Draw a line with endpoints at 2,2 and 6,6.
With the **ID** command and **END**point object snap, select the first endpoint of the line.
What coordinates does the command return?
Repeat for the other end. Are the coordinates what you expect?
2. Draw a line, circle, and arc on the screen.
Select the line with the **LIST** command.
What is the line's angle in the XY Plane?
Select the circle with **LIST** command.
What is the circle's radius?
Select the arc with **LIST** command.
What is the arc's starting angle?
3. Continue with the drawing from exercise #2.
Select the line with the **PROPERTIES** command.
Deselect the line, and then select the arc.
How do the data displayed by the Properties window change?
Select all three objects. Does the Properties window display a smaller or larger amount of information?
4. Continue with the same drawing, and enter the **DBLIST** command.
What do you notice about the information listed by AutoCAD?
Use the **COPY** command's **Multiple** option to copy the objects three times.
Use the **DBLIST** command again, and notice the length of the listing.
5. Continuing with the same drawing, use the **DIST** command with **END**point object snap.
Select two endpoints of one of the lines on the screen.
Is the length the same as reported by the **LIST** command?
Repeat the **DIST** command, but use **PER**pendicular object snap to find the distance between a line and a circle.
6. Draw two rectangles of different sizes with the **RECTANG** command.
Use the **AREA** command to determine the area of one rectangle.
Which option of the **AREA** command did you use?
Repeat the same procedure on the second rectangle. Add the two areas together, and obtain the total area.
Use the **AREA** command, but this time use the **Add** option to find the area of both rectangles.
Does AutoCAD arrive at the same answer as you?

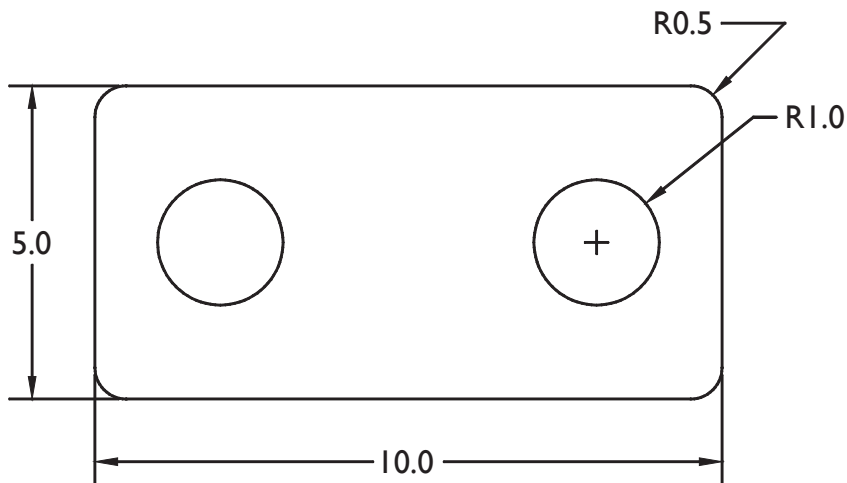
7. From the CD, open the *area.dwg* file, a drawing of geometric shapes. Find the area of each object, and then write down the figures. Using the **Add** option, find the total area of all objects.



8. Use the **TIME** command to check the current time in the drawing. Is the time reported by AutoCAD the same as on your wristwatch? The same as shown on your computer?
9. Use the elapsed timer option of the **TIME** command to determine the time needed to complete the following exercise. Before starting, write down the time you estimate it will take you to complete exercise #10.
10. Draw the baseplate shown below.

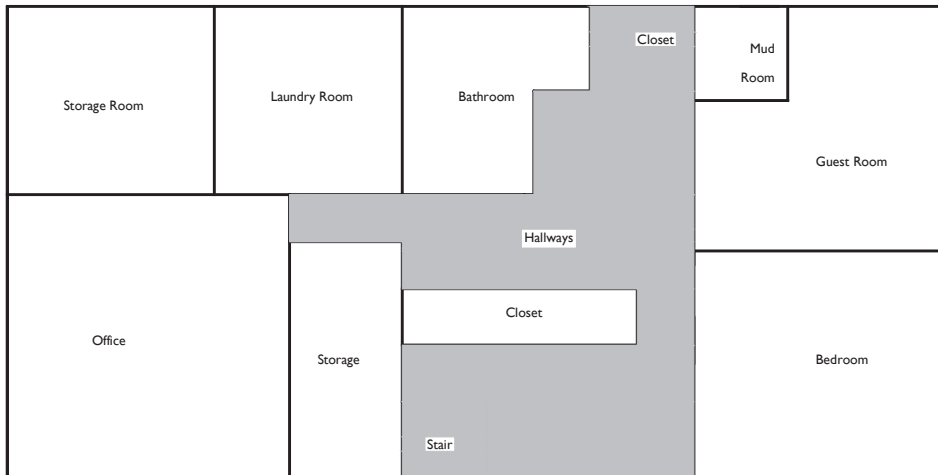
Use the **AREA** command to find the area.

Hints: Use the **RECTANGLE** command with its **Fillet** option. Remember to use the **Subtract** option to remove the holes from the total.



Turn off the timer. How close was your estimate?

11. From the CD, open the 34486.dwg file, a drawing of the author's 50' x 25' basement floor plan. Each unit in the drawing represents a foot.



What is the total square footage of the basement?

Create a new layer called "Areas."

Draw rectangles that match each room.

Find the area of each room:

Storage Room

Laundry Room

Bathroom

Mud Room

Guest Room

Bedroom

Closet

Storage

Office

What is the area of the hallways, including stairs?

12. Start a new drawing with *acad.dwt*.

Find the value of the following system variables:

OFFSETGAPTYPE

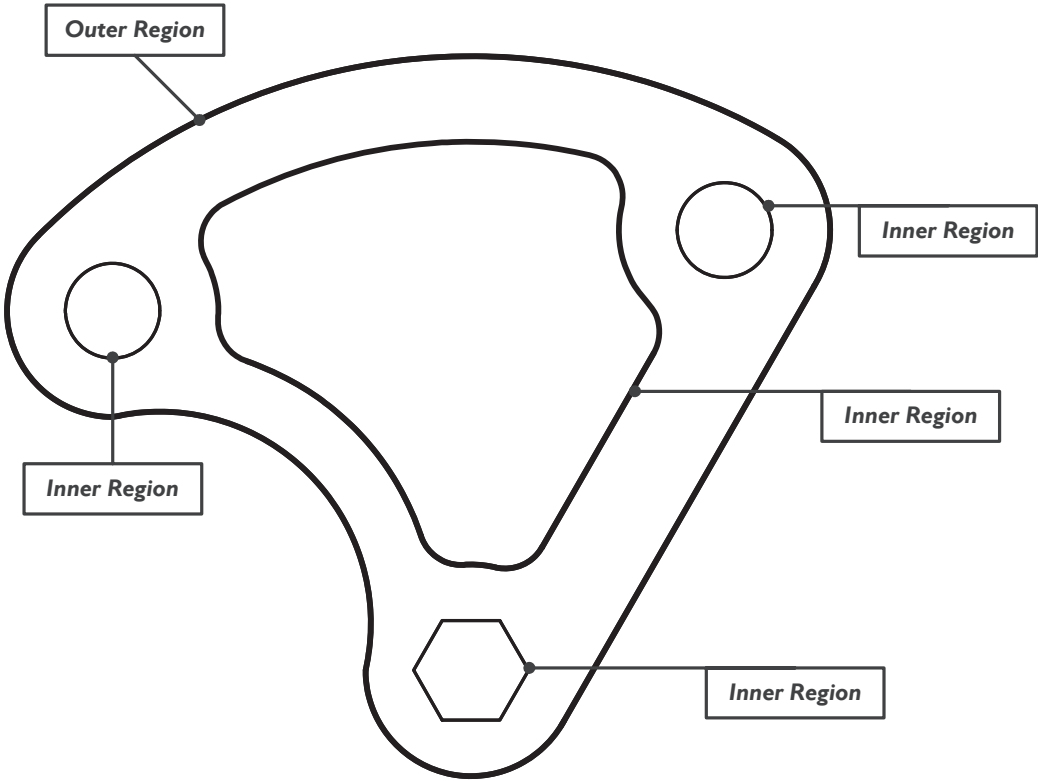
LASTPOINT

AREA

MAXSORT

If you do not understand the meaning of a system variable, refer to Appendix C of this book.

13. From the CD, open the *15_44.dwg* file, a drawing of a gasket.
- Ensure the **DELOBJ** system variable is set to 1.
- Start the **BOUNDARY** command, and set object type to **Region**. Click **Pick Points**, and select a point inside the gasket. How many regions is the gasket composed of?



Start the **SUBTRACT** command, and then select the outermost region. When prompted “Select solids and regions to subtract,” select the four inner regions (also called “islands”).

Apply the **MASSPROP** command to the gasket, and then save the data to an *.mpr* file. Write down the values from the report:

Area: _____

Perimeter: _____

Bounding box: X: _____ Y: _____

Centroid: X: _____ Y: _____

Moments of inertia: X: _____ Y: _____

Product of inertia: XY: _____

Radii of gyration: X: _____ Y: _____

Principal moments and X-Y directions about centroid: I: _____ J: _____

Use the **POINT** command to place a point at the centroid of the gasket.

14. From the CD, open the *I5_43.dwg* file, and find the centroid of the gasket.
15. Employ AutoCAD's **TABLE** command to create a table with seven rows and five columns.
16. Use the **TABLE** command to recreate the table of reference drawings illustrated below. (A copy of the table is available on the Companion CD in the *refdwg.tif* file.)

B-46858	CONV. C-12, LAYOUT
A-50818	CHUTE AT DISCHARGE END, ARRG'T & DET.
REFERENCE DRAWINGS	

17. With the **TABLE** command, recreate the table of general notes illustrated below. (A copy of the table is available on the Companion CD in the *generalnotes.tif* file.)

1.	ALL FIELD WELDED CONSTRUCTION (EXCEPT LADDER)
2.	PAINT NOTE: 1 COAT OF RED PRIMER AND 1 COAT OF EQUIPMENT GREY (SHOP PAINT LADDER, FIELD PAINT PLATFORM AFTER INSTALLATION)
3.	ALL WELDING MUST CONFORM TO THE REQUIREMENTS OF THE LATEST ISSUE OF C.S.A. STANDARD W.59 EXCEPT THE REFERENCE TO C.S.A. STANDARD W.47 DOES NOT APPLY.
GENERAL NOTES	

18. Recreate the bill of materials table illustrated below. (A copy of the table is available on the Companion CD in the *table-scan.tif* file.)

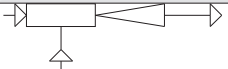
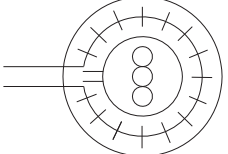
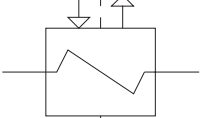
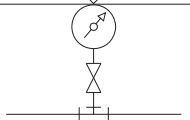
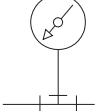
Use auto-fill to fill the cells under "No. PER UNIT."

BILL OF MATERIAL						
PART No.	No. OF UNITS	No. PER UNIT	DESCRIPTION	MATERIAL	STOCK No.	REMARKS
C-9248-1.	2		WASTE OIL DISPOSAL UNIT			
-2	226	2	3/16" PLATE x: 1'-1 1/2" Ø	STEEL	2502001	
-3	✓	2	3/16" x 3" FLAT BAR x 3'-6 1/2"	"	2500205	
-4	✓	2	3" x 3" BUTT HINGE	"	1970323	
-5	✓	2	3/8" x 1 1/2" FLAT BAR x 3"	"	2500503	
-6	✓	2	12" STD. BLACK PIPE x 1'-6"	"	3010003	
-7	✓	2	24 MESH SCREEN, 11-3/4" Ø	S.S.	2040201	
-8	✓	2	3/8" ROUND x 3'-0" LG.	STEEL	2501703	

19. From the Companion CD, open the *chapter16.dwg* file, a drawing of a parts list table.
- a. Edit the table to add the values in the **Qty** column (short for “quantity”). What approach did you take to enter the values?

Parts List			
Item	Qty	Part Number	Description
1	5	PC-15	Paper clips
2	4	P-24	Pencils
3	3	PP-33	Paper Pads
4	2	R-42	Rulers
5	1	CB-51	Clipboard

- b. Split the table into two sections.
20. Create the parts list table illustrated below. In the **Description** column, insert the appropriate blocks. Describe the approach you took to insert the blocks.

Parts List			
Item	Qty	Part	Illustration
1	5	Air Injector (Dynamic Pump)	
2	4	Evaporator - Circular Fin	
3	3	Heater Feed with Air Outlet	
4	2	Pressure Gauge	
5	1	Thermometer - Type 2	

21. From the Companion CD, open the *table.dwg* file, and then use the **TABLEEXPORT** command to create a .csv file.
- If you have access to a spreadsheet program, open the .csv file and view its contents.
- Is the data in the spreadsheet the same as the table in AutoCAD?
22. From the Companion CD, open the *tablestyle.dwg* file.
- Use the **TABLESTYLE** command to make a new style based on the Standard style:
- a. Change font to Times New Roman.
 - b. Change fill color of data rows to light gray.
 - c. Change border line color to white.
 - d. Name the style.

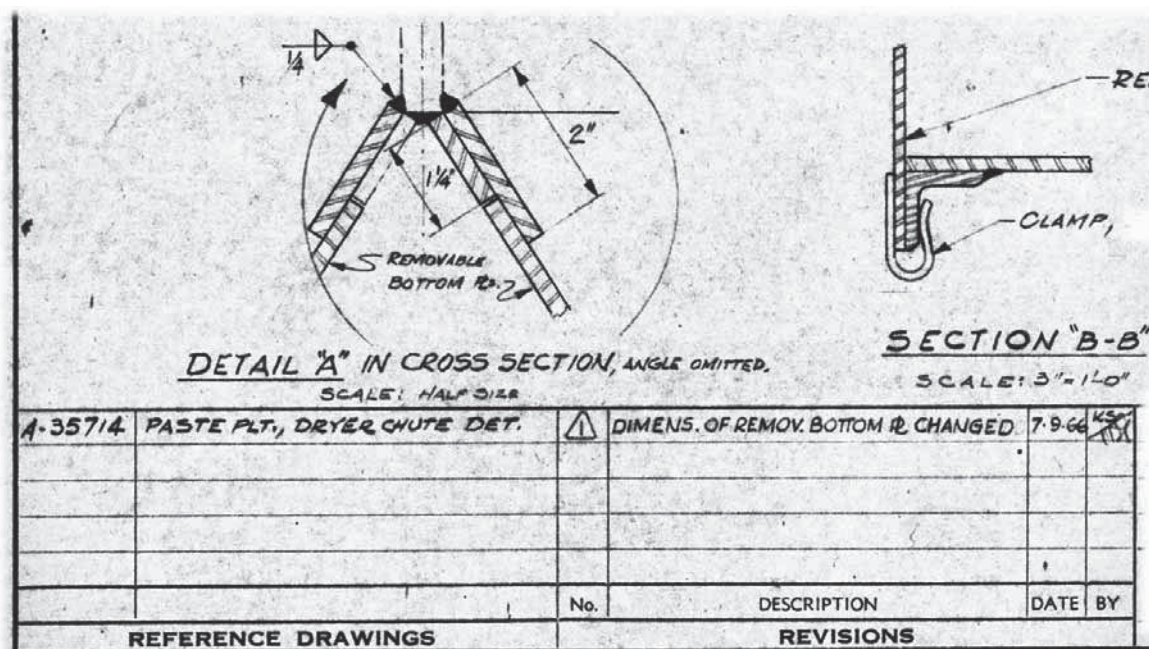
CHAPTER REVIEW

1. List the two kinds of actions the **ID** command performs:
 - a.
 - b.
2. Where are the **ID** coordinates stored?
How do you access them in other drawing and editing commands?
3. What is the difference among the **PROPERTIES** and **LIST** and the **DBLIST** commands?
4. What are the six distances returned by the **MEASUREGEOM** command?
5. What option of the **MEASUREGEOM** command's **Area** option calculates the area of a circle?
6. After using the **MEASUREGEOM** and **LIST** commands on the circle, which command do you find more useful?
7. How do you determine the last time a drawing was updated?
8. Can the **MEASUREGEOM** command return the length of an open polyline?
9. Why would you want to subtract areas?
10. Describe a purpose of the **DWGPROPS** dialog box.
11. List two reasons why the Total Editing Time might be inaccurate for drawings:
 - a.
 - b.
12. Can you change the value of all system variables?
13. Examine the bill of material below, and then answer the following questions. (Also available as *bofm.tif* on the Companion CD.)

BILL OF MATERIAL						
PART No.	No. OF UNITS	No. PER UNIT	DESCRIPTION	MATERIAL	REQ. No.	REMARKS
C-10102-1	1		LUB. DRIP TRAY ASSY.			
-2		1	16 GA. x 1'-8 1/2" x 7'-3" SH. MET. TRAY	M. S.	2503101	
-3		1	2" Ø STD. PIPE NIPPLE, 4" LG.	"	3252405	
-4		1	1/4" N.C. HX. HD. BOLT, 1" LG.	"	DE-CONTE	TACK-WELDED TO TRAY
-5		2	5/16" Ø FLAT WASHER	"	-8-	ONE AS SHOWN
-6		1	1/4" N.C. WING NUT	"	2320501	
-8		1	3/8" x 2" x 2" L, 2'-0" LG	M.S.	2501300	
-9		1	3/16" x 1 1/2" x 1 1/2" L, 8'-4" LG	"	2501002	
-10		1	-8-, 10" LG	"	-8-	

- a. How many columns does the table have?
- b. How many rows?
- c. What is the wording of the title row?
- d. What does AutoCAD call the row containing the phrase "PART No."?
- e. Where is the insertion point located?

14. Examine the bill of material below, and then answer the following questions. (Also available as *revisions.tif* on the Companion CD.)



- a. How many columns does the table have?
 - b. How many rows?
 - c. Is the table's direction up or down?
 - d. What might be the purpose of the Reference Drawings table?
Of the Revisions table?
15. Explain the reason behind tables having title, header, and data rows.
 16. Can individual cells be formatted differently from each other?
 17. Where is the title row located when the table's direction is up?
 18. After creating a table, can rows be added?
Removed?



3D Modeling

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CHAPTER 16

Introduction to 3D Solid Modeling

In 2D drafting, you draw and edit with 2D objects — such as lines, arcs, and circles — to construct drawings. In 3D modeling, the process is similar: you draw and edit with basic boxes, spheres, and other 3D objects to produce complete drawings.

The difference between 2D and 3D lies in the nature of the basic objects, the kinds of editing tools available, and the added dimension (z, or depth). Editing consists of adding, subtracting, and manipulating solids. For example, holes are created by subtracting cylinders from models.

Solid modeling is used primarily in mechanical engineering, because models are precise and can be accurately analyzed. Solid modeling is also sometimes used by architects to represent complex intersections, such as the roofs on buildings, and to find interferences, such as between plumbing and the building's walls.

After constructing 3D models, designers often “convert” them to standard sets of 2D drawings for construction. The word “model” is often used to distinguish 3D drawings from 2D ones.

In this chapter, you learn the following commands for constructing solid models:

BOX creates basic 3D solid models.

PRESSPULL extrudes 2D objects into 3D models.

REVOLVE creates round solid bodies from 2D designs.

UNION joins two or more 3D solids into bodies.

FILLET rounds off edges of 3D solids.

Subobject selection selects vertices, edges, faces, and subobjects.

ViewCube rotates 3D viewpoints in real time.

UCS and **Dynamic UCS** change the orientation of the x, y workplane.

HISTORY OF 3D IN AUTOCAD

AutoCAD provides several methods of creating 3D objects, each of which has its advantages and disadvantages. Historically, each method was added to AutoCAD as subsequent releases improved their ability to work in three dimensions.

It is rumored that the very first release of AutoCAD contained 3D, but a dispute with the programmer led to the feature remaining unusable. Over the following decades, Autodesk added new 3D technologies frequently, such as polyfaces (useful for surveying and digital terrain modeling), solid modeling (useful for mechanical designs), surfaces (useful for conceptual modeling in architecture), and now mesh modeling (useful for industrial design). The catch to this ad-hoc development is that older commands that seemed to create surfaces and meshes, don't — such as **TABSURF** and **3DMESH**.

Thickness. AutoCAD v2.1 gave certain 2D objects thickness and elevation using the **ELEV** command. This is sometimes called “2-1/2D,” because the extrusions always result in flat tops. Adding thickness is easy to understand and edit, but is limited in capability, and sometimes renders incorrectly.

Wireframes. AutoCAD Release 9 created 3D wireframe objects with the **3DLINE** command; later the **3DPOLY** command was added, and many editing commands were extended to allow 3D editing. Wireframe objects, however, are difficult to construct, are even more difficult to edit, and do not render at all.

Surface meshes. As far back as 1989, Autodesk announced that Release 10 was “the 3D release,” because it created 3D objects from polyface meshes with commands like **3DMESH**, **RULESURF**, and **3D**. *Polyface meshes* are triangular and rectangular planes that are joined to create 3D surfaces. They are difficult to edit, and AutoCAD provides little assistance in manipulating surfaces.

Solid modeling. AutoCAD Release 11 created 3D objects from solid models with commands such as **BOX**, **EXTRUDE**, and **SOLIDEDIT**. Over time, Autodesk switched the solid modeling engine in AutoCAD from **PADL** to **ACIS**, and now to **ShapeManager**. Solids are popular for designing mechanical devices. In this chapter, you work with solid models.

Surface modeling. AutoCAD 2007 added commands to create surfaces, such as **LOFT** and **SWEEP**. These are “true” surfaces, and not the simulated ones created by polyface meshes.

Mesh modeling. AutoCAD 2010 added 3D meshes, which are popular for styling automobiles and consumer products. To distinguish them from pre-2010 3D mesh objects made of polyfaces, Autodesk calls the new ones “full featured mesh objects” and “subdivision”; in this book, they are named “3D mesh objects.”

Custom 3D objects. AutoCAD Release 13's ObjectARx programming interface made it possible for Autodesk and third-party developers to create custom 3D objects, such as the landscape objects created by the **LSNEW** command (since removed from AutoCAD).

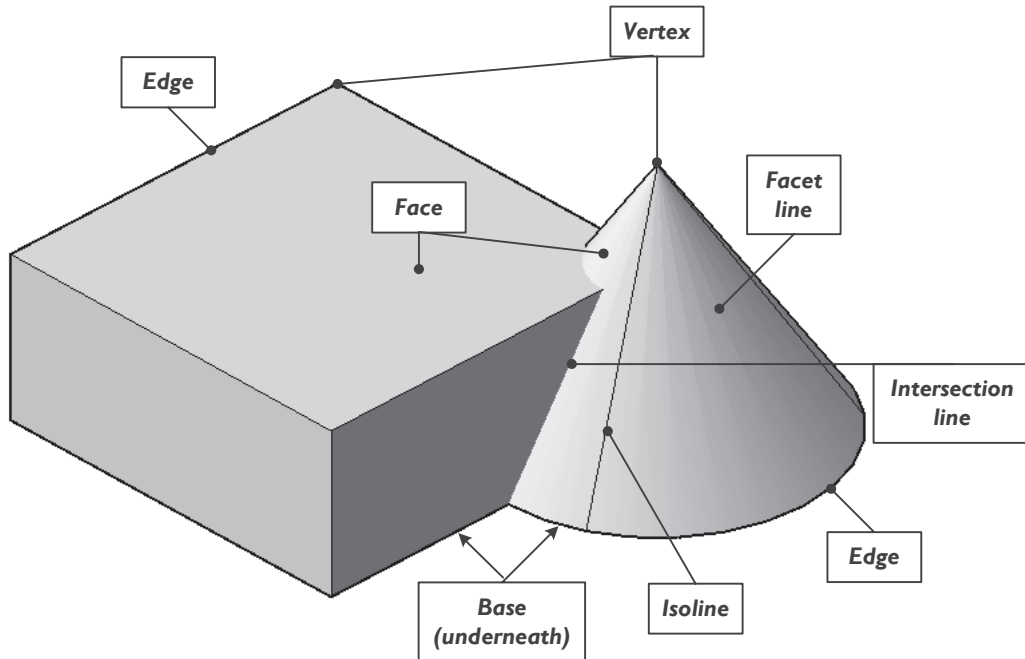
Photorealism. AutoCAD Release 12 added the **RENDER**, **LIGHT**, **RMAT**, and related commands for creating realistic images of 3D models. This was improved in AutoCAD 2007 with improved photorealism and material definitions.

Visual styles. AutoCAD 2007 added visual styles that allowed users to edit 3D drawings in simulated rendering modes. Visual styles can be customized to output drawings that are suitable for technical documentation or for appearing hand-sketched.

In this book, you learn about 3D solid modeling (this chapter) and 3D mesh modeling (the following chapter). Solid modeling is better for precise prismatic shapes, such as machine parts, while 3D meshes are better for organic flowing shapes, such as hair dryers and cell phones.

PARTS OF 3D SOLID MODELS

Three-dimensional models have an entire vocabulary to define features (a.k.a. *subobjects*), as illustrated below.



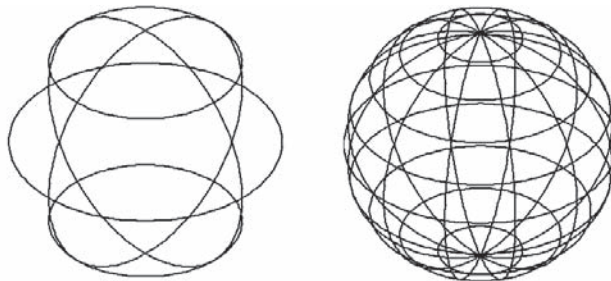
Faces define the surfaces of solid models. Faces can be flat (as on boxes) or curved, as on spheres and cones. When drawings are in 2D or 3D wireframe mode, faces are invisible.

Edges outline faces. On boxes, each face has four edges; a cone has just one edge, around the base. The look of edges can be modified with the `VISUALSTYLE` command.

Vertices are at the intersections of edges and faces. On boxes, a vertex is located at each corner, eight in all. Cones have one vertex, at the tip.

Bases are the bottoms of 3D models. When creating primitives like boxes and cylinders, you first define the size of the base, and then the height. The bases of boxes are squares or rectangles; the bases of cone are circles or ellipses.

Isolines illustrate the surface of curved 3D models, such as spheres and cones. Since curved surfaces have no edges, isolines help you visualize the curves. The number of isolines is adjusted with the `ISOLINES` system variable; the default is 4 isolines, but a better value is 12.



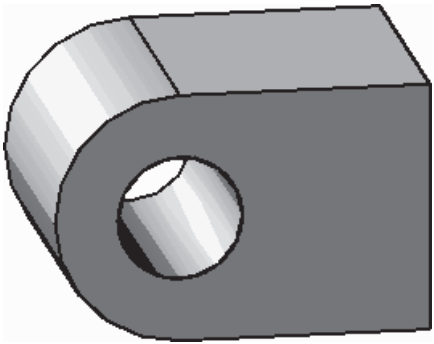
Left to right: `Isolines = 0` (sphere is invisible), 4 (default value), and 12.

Facet lines appear when curved portions of 3D models are not smoothly rendered. Unsmooth renderings are faster than smoothed renderings.

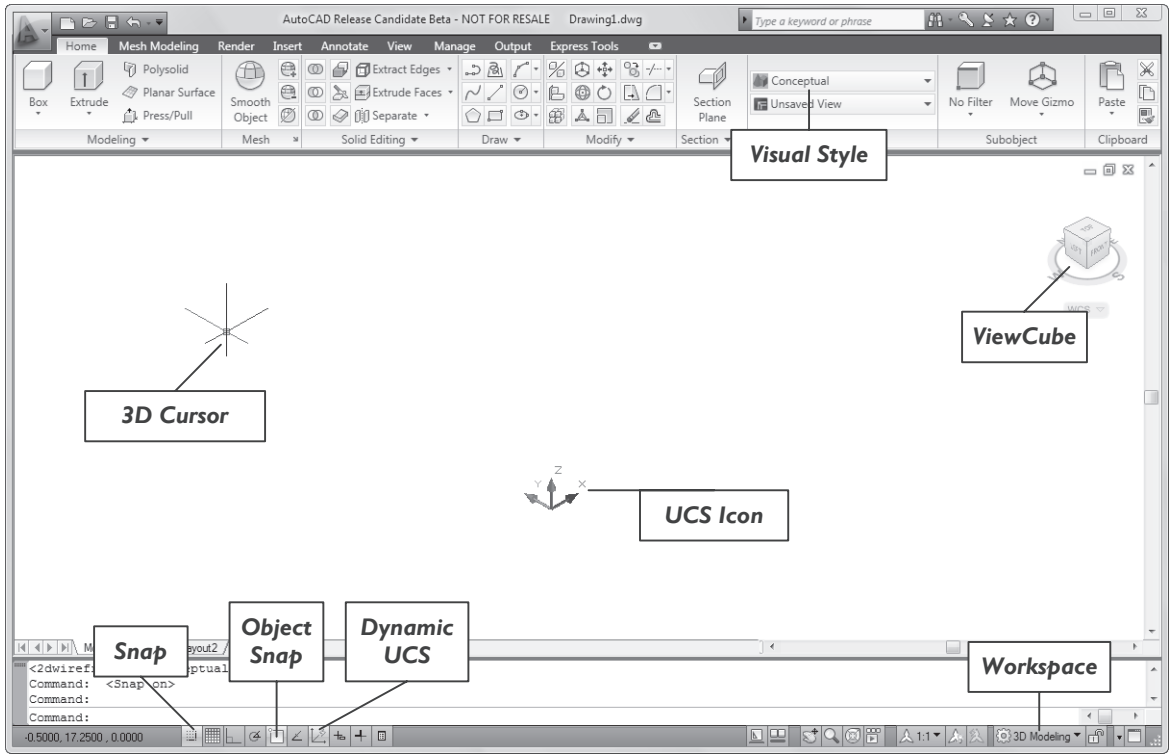
Intersection lines appear at the intersections of 3D models.

TUTORIAL: PREPARING TO MODEL IN THREE DIMENSIONS

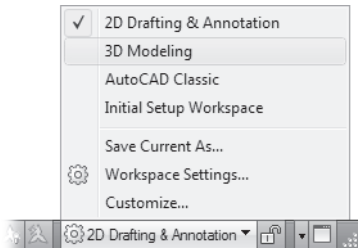
In this tutorial, you prepare AutoCAD to model the simple part in 3D, as illustrated below.



Start AutoCAD with a new drawing (*acad3d.dwt*), and then follow these steps. (See figure for the locations of options.)



- I. Set the workspace for 3D modeling, as follows:
 - a. On the status bar, click the **Workspace** droplist.
 - b. Choose “3D Modeling.”



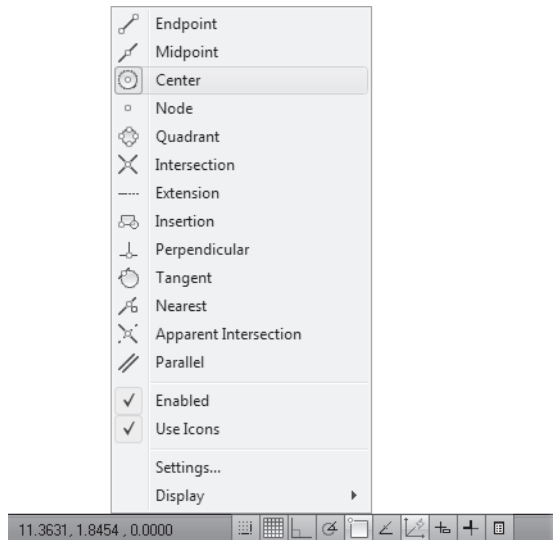
Notice that the ribbon changes to emphasize 3D drawing and editing tasks. (If the Tools palette appears, close it.)

2. Even though AutoCAD is a 3D CAD program, much “3D” drawing is actually done in 2D on the x, y -plane. The *dynamic UCS* (user-defined coordinate system) causes AutoCAD to automatically rotate the x, y-plane to match the selected face, as you will see later in this tutorial. (More on UCSs later in this chapter.)

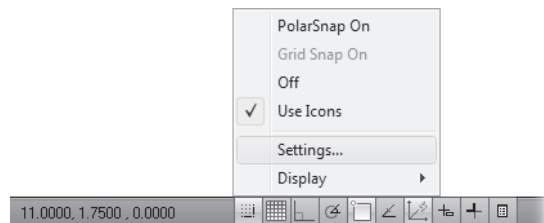
Ensure that the **Allow Dynamic Ucs** button is turned on: on the status bar, the button should look blue.



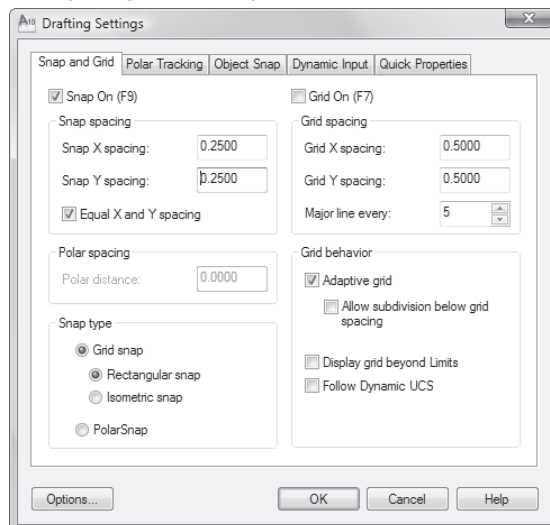
3. Ensure that **Object Snaps** is turned on and that **Center** is the only snap mode.



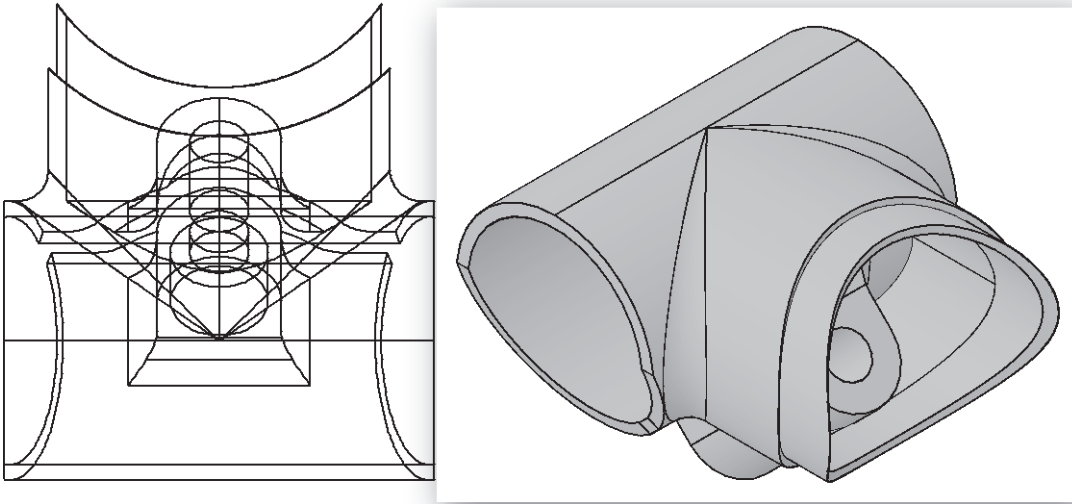
4. Turn on **Snap** mode, and then set the snap spacing to the smallest distance in this model, **0.25**, as follows:



- a. Right-click the **Snap** button, and then choose **Settings** from the shortcut menu.
- b. Enter **0.25** for the X Spacing, and then press **Tab**.



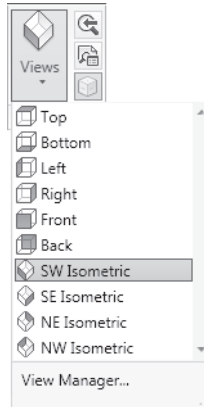
- c. Click **OK**.
5. Change the viewpoint from plan view to isometric view. This viewpoint lets you see three sides of the model at the same time: front, side, and top.



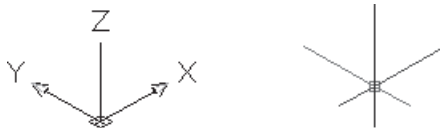
Left: A wireframe 3D model in plan view.

Right: Same model viewed from an isometric viewpoint in Conceptual rendering.

- a. Click the ribbon's **View** tab.
- b. In the **Views** panel, click the **Views** button, and then choose "SW isometric."



Notice that the cursor and UCS icon rotate to make all three axes (x, y, and z) visible.

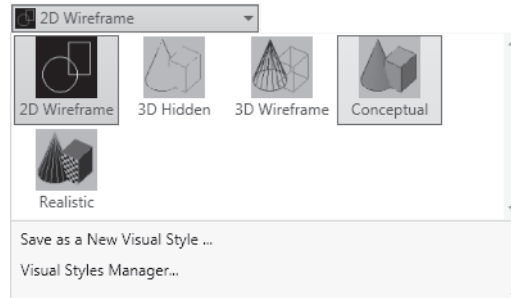


Left: UCS icon shows x, y, and z axes.

Right: 3D cursor.

6. Change the visual style to Conceptual. This is a rendering mode (also known as a "visual style") that makes it easier to view the 3D model than it would be in traditional CAD wireframe mode. (See earlier figure.)
 - a. Click the ribbon's **Home** tab.

- b. In the **View** panel, click the **2D Wireframe** droplist, and then choose “Conceptual.”

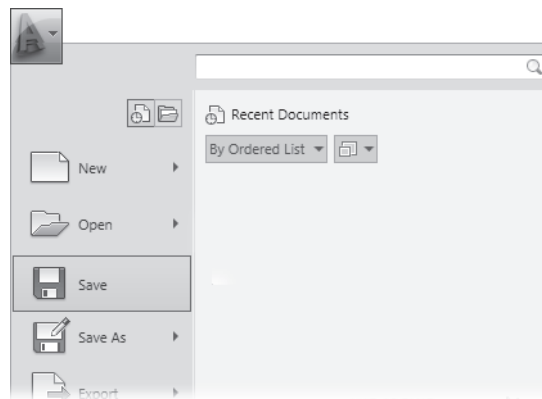


Notice the drawing area again changes: the UCS icon changes to color and the ViewCube appears in the upper right corner; these two are useful for orienting yourself in 3D space and for quickly changing the 3D viewpoint.



Left: UCS icon with a different color for each axis.
Right: ViewCube.

7. With this preliminary work accomplished, save the drawing, as follows:
 - a. Click the red **A**.
 - b. Choose **Save**.



- c. In the Save Drawing As dialog box, name the drawing “First 3D Model.”
 - d. Click **Save**.

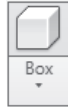
TUTORIAL: DRAWING SOLID PRIMITIVES

When creating 3D drawings, you usually construct them in one of two ways:

- From basic 3D elements called “primitives” like boxes, cones, and spheres.
- By turning 2D elements like circles and polygons into 3D elements through actions like extruding and revolving.

In this part of the tutorial, you draw a 3D box shape; later, you turn a circle into a cylinder.

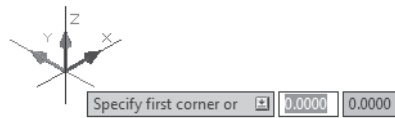
1. In the Home tab's Modeling panel, chose **Box**.



Notice that AutoCAD prompts you in the command bar. If dynamic input is turned on, you also see the prompts in the drawing area.

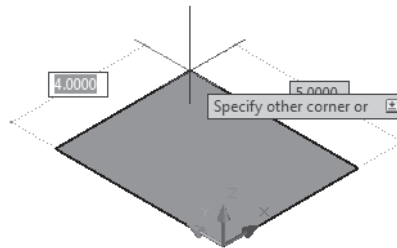
Boxes are defined by two corners of their rectangular base and their height.

2. Start the box at the origin of the drawing, which is indicated by the UCS icon:
Specify corner of box or [CEnter] <0,0,0>: (Pick point 1.)

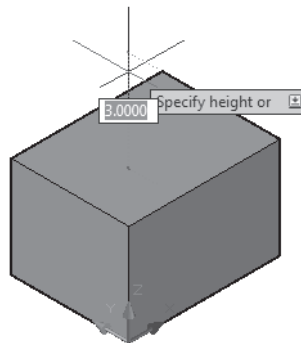


Recall that the origin has x, y coordinates of 0,0. (With snap mode is turned on, it is easy to pick the 0,0 point precisely.)

3. For the opposite corner of the box's rectangular base, drag the preview rectangle to 5,4:
Specify corner or [Cube/Length]: (Pick point 2.)



4. For the height, drag the rectangle to 3.
Specify height: (Drag the box to a height of 3.)



Specifying the height is the same as specifying the z coordinate.

5. You have drawn a box as a 3D solid model. Press **CTRL+S** to save your work.

If you find it too finicky to show AutoCAD the size of the box, you can enter numbers for the x,y coordinates and the height like this:

Specify corner of box or [CEnter] <0,0,0>: **0,0**

Specify corner or [Cube/Length]: **5,4**

Specify height: **3**

Other commands for creating 3D primitives include **SPHERE**, **TORUS** (donut-like shapes), **WEDGE**, **CYLINDER**, and **CONE**. You can practice them in the exercises at the end of this chapter.



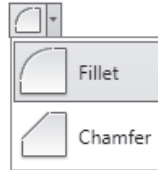
Note To zoom the 3D model, place the cursor over the 3D model, and then roll the mouse's roller wheel. In one direction, the model zooms larger; in the other, smaller. By having the cursor over the model, you keep it centered in the drawing area while zooming.

TUTORIAL: MODIFYING 3D MODELS

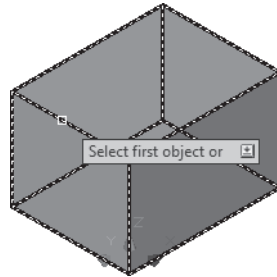
Once a basic 3D model is drawn, you can modify it either with generic editing commands, like **FILLET** and **CHAMFER**, or with commands specific to 3D, such as **SOLIDEDIT** and **PRESSPULL**. You can also edit 3D models directly with grips, but in a more limited fashion than with 2D objects.

In this tutorial, you round one end of the box with the **FILLET** command. The command operates in 3D somewhat differently from what you may be used to in 2D.

1. In the Home tab's Modify panel, choose **Fillet**.

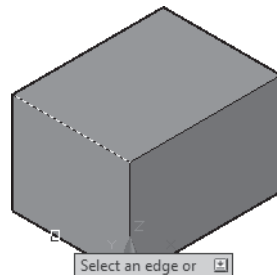


2. When the AutoCAD prompts you to select an object, click the edge shown in the figure:
Select first object or [Undo/Polyline/Radius/Trim/Multiple]: *(Pick the edge.)*

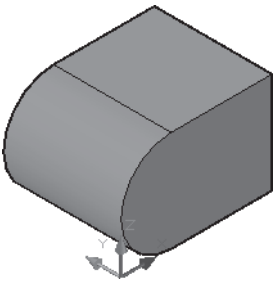


3. Specify the radius of 1.5 units (half of the box's height of 3 units):
Enter fillet radius: **1.5**

4. Select the lower edge, as shown below:
Select an edge or [Chain/Radius]: *(Pick lower edge.)*



- 5. Press **ENTER** to end the command:
Select an edge or [Chain/Radius]: *(Press Enter.)*
Notice that the one end is instantly redrawn with curvature.

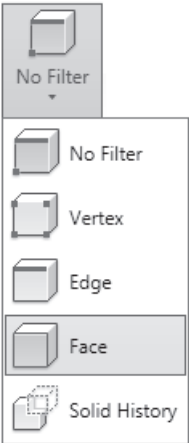


TUTORIAL: CREATING HOLES

One of the most common actions in 3D modeling is removing holes. This is done indirectly: you remove a cylinder shape from the model to create the hole. The action is done two steps: (1) draw a cylinder, and then (2) remove it from the model.

You could use the **CYLINDER** command, but I find it easier to place a circle, and then extrude it into a cylinder shape. This is an example of the second method of creating 3D models: extruding 2D objects.

- 1. You will be placing the circle on a side face of the now-rounded box. Selecting faces can be tricky, and so AutoCAD provides subobject filtering. This makes it easier to select a specific part of a 3D object, such as a single face or edge.
In the Home tab's Subobject panel, click **No Filter**, and then choose **Face**.



- 2. Hold down **CTRL** Key. Notice the square face icon near the cursor. It indicates that subselection mode is set to Face.
The complete list of subobject filters is shown in the following table:

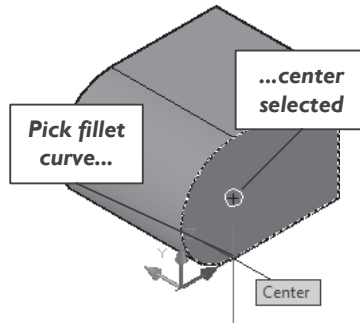
Subobject Filter	Icon	Meaning
None		Selects the 3D model.
Vertex		Selects a vertex.
Edge		Selects an edge.
Face		Selects a face.
Solid History		Selects a primitive used to make the 3D model.

3. In the Home tab's Draw palette, choose **Circle**.



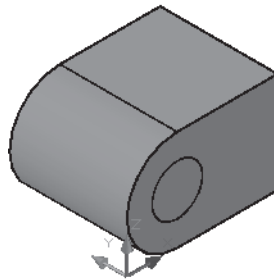
4. Move cursor near the semicircle of the fillet; notice that AutoCAD's object snap finds the center of the fillet.

Specify center point for circle or [3P/2P/Ttr (tan tan radius)]: *(Pick the filleted curve.)*



5. Enter the radius of 0.75 units:

Specify radius of circle or [Diameter]: **.75**



6. With the circle drawn, you extrude it into a cylinder shape. In the Home tab's Modeling panel, choose **Press/Pull**.



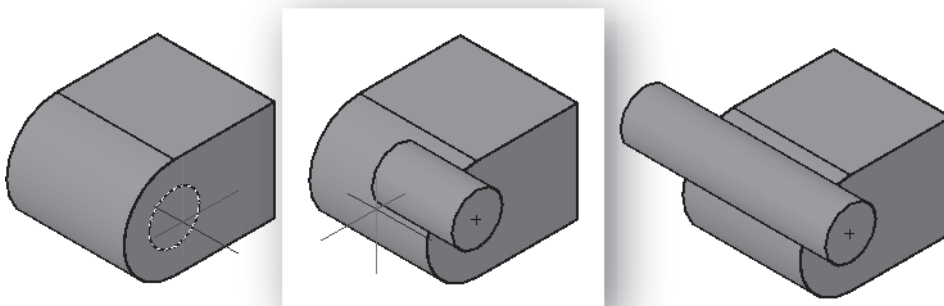
7. Turn the circle into a cylinder like this:

- a. Click inside the circle, and then drag it.

Notice that the 2D circle elongates into a 3D cylinder.

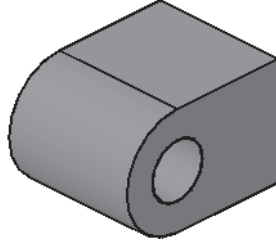
- b. Drag the length of the cylinder until it clears the other end of the model; the precise length does not matter.

- c. Click to end.

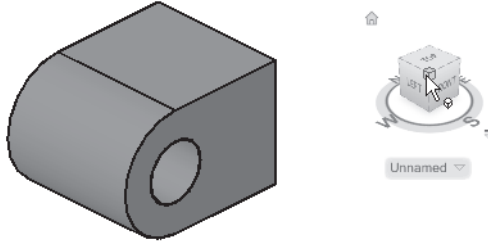


Left to right: Select the circle, and then drag it.

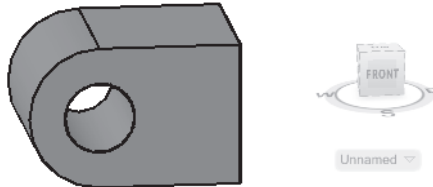
Notice that the model now has a hole through it.



8. You can get a better view of the hole by rotating the model like this:
 - a. Grab any part of the ViewCube.



- b. Drag the cube. As you do, notice that the 3D model rotates.



9. Press **CTRL+S** to save your work.

In this tutorial, you completed the most common solid modeling tasks:

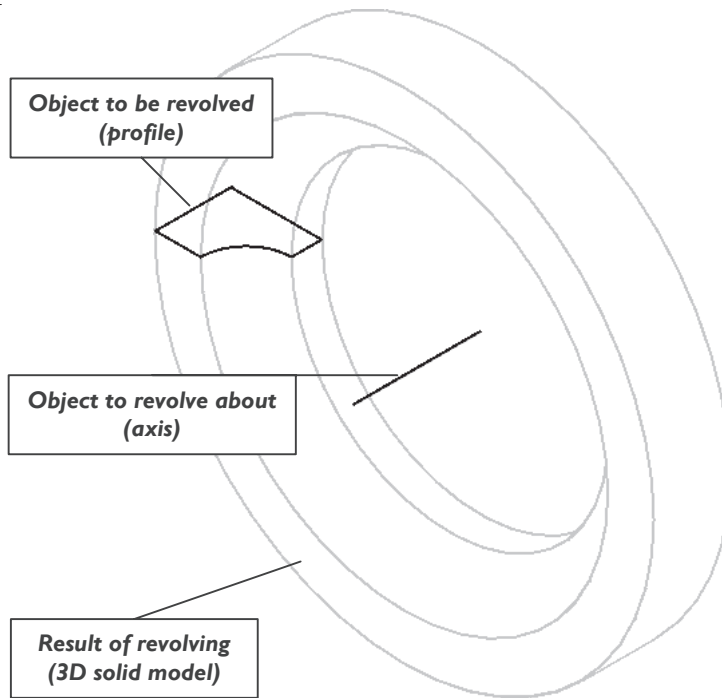
- Creating a 3D primitive (box).
- Modifying the primitive (fillet).
- Extruding a 3D shape from a 2D object (circle to cylinder).
- Rotating the 3D viewpoint (view cube).

REVOLVED MODELS

In the earlier tutorial, you created a 3D object (the cylinder) by extruding a circle. A second way to generate a 3D object is to revolve a 2D outline. The **REVOLVE** command creates symmetrically round objects, like wineglasses and chair legs.

To revolve (convert) 2D objects into 3D solids, AutoCAD needs to know two things: (1) the objects to revolve, and (2) the axis about which to revolve them.

Objects that can be revolved include circles, ellipses, closed polylines, polygons, donuts, closed splines, and regions. (AutoCAD cannot revolve objects within blocks, or polylines that cross over themselves.) These objects are known as the *profile*.

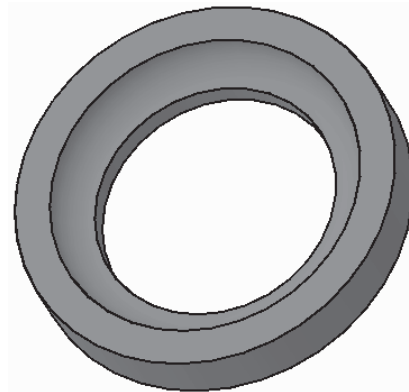


The axis can be the x or y axes, or an object, such as a line or polyline segment. The length of the axis is unimportant; AutoCAD uses only the axis' orientation in space to determine the location and angle of the revolution.

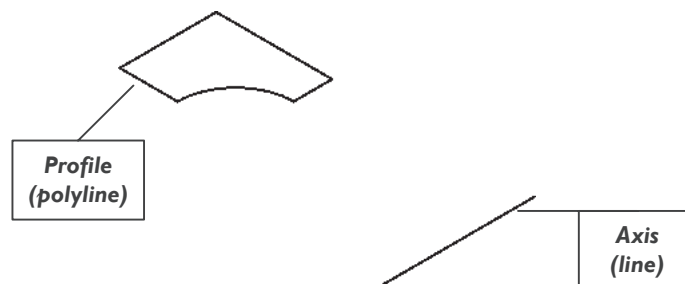
You can specify full 360-degree revolutions, or partial revolutions, with positive or negative angles.

TUTORIAL: REVOLVING OBJECTS

In this tutorial, you create a small washer in 3D. Since it is *round*, it makes sense to create it by *revolving* the profile of its cross-section. The result of the tutorial is illustrated below.

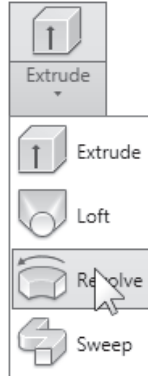


- I. From the Companion CD, open the *3D washer.dwg* file, the drawing of a rubber washer's profile. This washer design has an inner fillet.

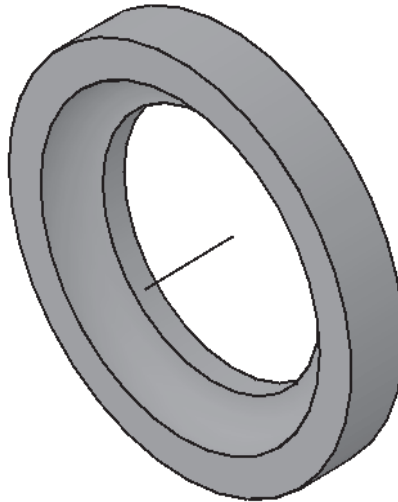


Notice that the profile is drawn from a polyline; the single line is the axis about which the profile will be revolved.

2. In the Home tab's Modeling panel, click **Extrude**, and then choose **Revolve**.

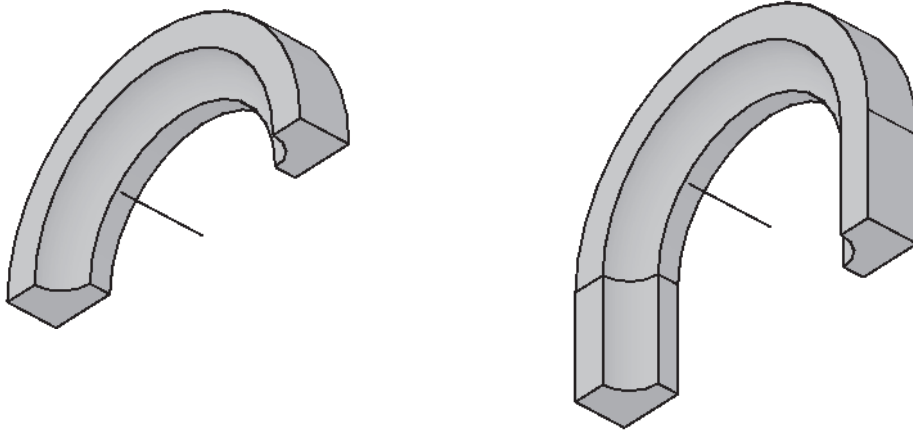


3. Select the profile (polyline outline), and then press **ENTER**:
 Select objects to revolve: *(Choose the profile.)*
 Select objects to revolve: *(Press **Enter**.)*
4. Select the axis (line object):
 Specify axis start point or define axis by [Object/X/Y/Z] <Object>: **o**
 Select an object: *(Choose the line.)*
5. Finally, specify the angle of revolution. For a full washer, specify 360 degrees:
 Specify angle of revolution or [STart angle] <360>: *(Press **Enter**.)*
 Notice that AutoCAD instantly draws the washer in 3D as a solid model.



6. Erase the line: select it, and then press **DEL**.

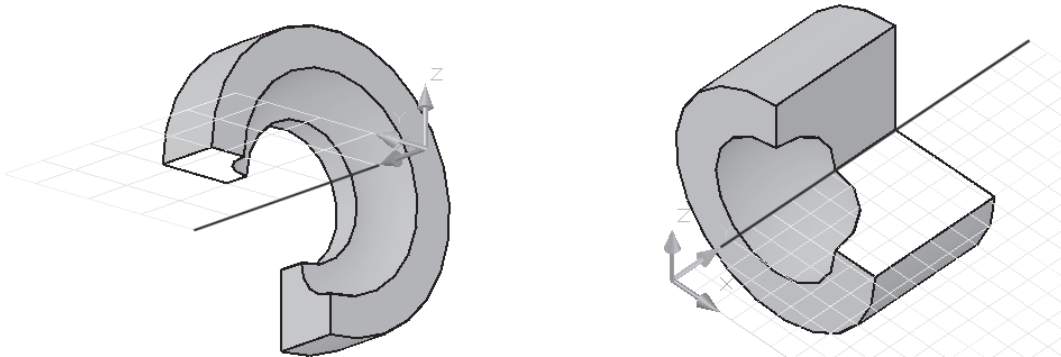
If you specify a smaller angle of revolution, you get a washer that's been cut (at left). You can use the **PRESSPULL** command to extend parts of the washer (at right) to create a horseshoe-like shape.



Left: Profile revolved by 180 degrees.

Right: Endcaps of the 3D model extruded.

You get different results, depending on the axis you choose. The figures below show the result of revolving the same profile about the x and y axes — instead of the line. The resulting shape differs, as does the size. (I revolved them by 270 degrees so that you can see the profile, and included the UCS icon so that you can see the axis; I colored the area of the profile with white.) This shows that the distance of the profile from the axis affects the shape and size, and so it is crucial to place the profile correctly relative to the axis.



Left: Profile revolved about x axis.

Right: Profile revolved about y axis; washer hole is closed.

UNDERSTANDING THE UCS

To create and edit 3D drawings, you must become comfortable with changing the point of view, and working at any angle in 3D space. AutoCAD provides a number of tools that create and save 3D viewpoints. One of the commands is UCS (short for user-defined coordinate system).

WORLD COORDINATE SYSTEM (WCS)

If there are coordinate systems that can be defined by users, then there must be another, absolute coordinate system to which they refer. There is: it is the “world coordinate system” (WCS, for short).

When you start AutoCAD with a template file like *acad.dwt*, it begins in WCS plan mode. You look straight down onto the x, y-plane, with the z axis coming out of the monitor towards your face. You can change the viewpoint, and it is still the WCS.

User-defined Coordinate Systems (UCS)

But when you change the coordinate system, you define a new one — a UCS. Why would you define new coordinate systems? To solve a problem that AutoCAD has; namely, it is so 2D-oriented that drawing and editing in 3D are hard.

The workaround is to create UCSes that rotate the x, y-plane in 3D space to match the plane in which you want to draw and edit. Think about drawing on the face of a sloped roof: the task is hard if you could not reorient the drawing plane to match the roof. The tutorial later in this section does exactly that.

UCS Icons

The UCS icon helps orient you in 3D space. When you first start AutoCAD, it shows the direction of the x and y axes in the WCS:

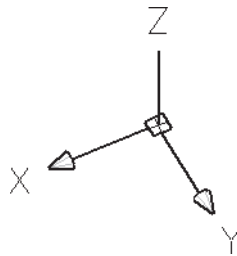


Left: UCS icon in WCS.

Right: UCS icon in UCS.

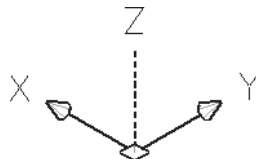
The small square at the intersection of the axes means that AutoCAD is in WCS (world coordinate system). You are looking straight down on the x,y-plane, so you do not see the z axis. When you invoke a user-defined coordinate system, the UCS icon loses the square.

When you change to a 3D viewpoint, however, the z axis appears. For example, enter **VPOINT 1,2,3** to see the following:

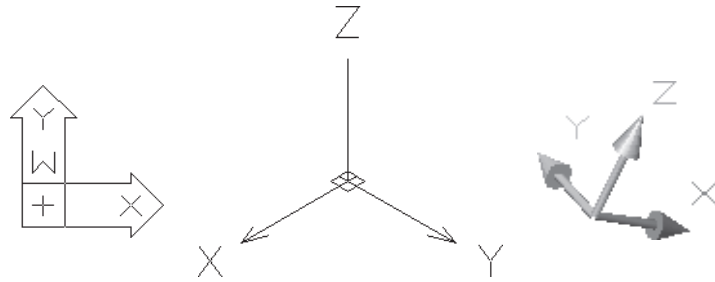


Look closely at the square and you will see a plus sign. This indicates that the UCS icon is located at the origin (0,0); when the plus sign is missing, the UCS icon is located elsewhere, typically in the lower left corner of the viewport.

When the viewpoint goes underneath the x, y-plane, the z axis is negative. The z portion of the UCS icon becomes dotted. Enter **VPOINT 1,1,-1** to see an example of this:



The icon can change its look. The original UCS icon showed only the direction of the x and y axes, and no z axis. It was introduced in 1990 when “full 3D” was added to AutoCAD Release 10. (See figure below.)



Left: Original 2D UCS icon first introduced to AutoCAD Release 10.

Center: 3D UCS icon displayed in wireframe mode.

Right: 3D UCS icon displayed by visual styles.

The icon illustrated in the center (above) is today's default icon in 2D wireframe model space. The icon is displayed to the right when the viewport is in a visual style mode, such as Conceptual.

When you switch to layouts, the icon is strictly 2D and strictly ornamental. There is no 3D in layouts, because they are meant to represent flat sheets of paper.

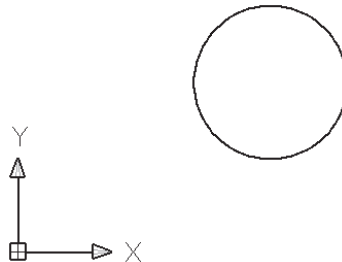


You can adjust the look of the UCS icon with the UCSICON command's **Properties** option.

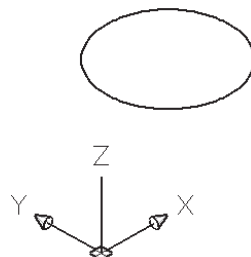
TUTORIAL: CHANGING FROM WCS TO UCS

In this tutorial, you draw an object in the WCS, and then draw the same object in the UCS to see how the two coordinate systems affect 2D drawing and editing in 3D space.

1. Start AutoCAD with a new drawing using the *acad.dwt* template file. AutoCAD always starts in WCS mode, the world coordinate system.
2. Draw a circle with a radius of 2. The circle is drawn in the x, y-plane of the WCS.



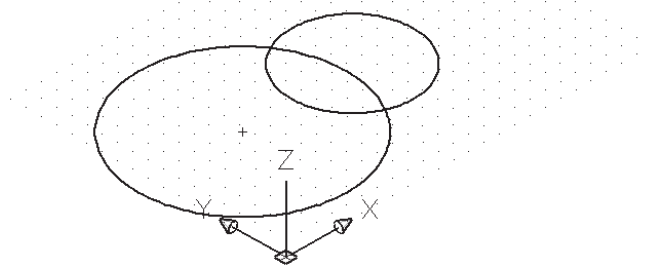
3. Rotate the 3D viewport: from the View tab's Views panel, choose **SW Isometric**. Notice that the circle and UCS icon rotate to match the new viewpoint; the circle looks like an ellipse because the viewpoint is rotated.



4. The drawing is still in WCS; only the viewpoint has changed (to -1,-1,1). To see this, draw another circle with any size of radius.

Also, turn on the grid, which always lays in the x,y-plane of the current UCS (or WCS, in this case).

Notice that the second circle and the grid lie in the same plane as the first circle.



5. New UCSs are created with the **UCS** command. In the View tab's **Coordinates** panel, choose  **UCS**. (Alternatively, enter **UCS** at the 'Command:' prompt.)

6. Enter **n** to create a new UCS:

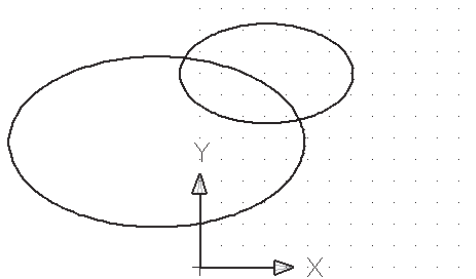
Enter an option

[New/Move/orthoGraphic/Prev/Restore/Save/Del/Apply/?/World] <World>: **n**

7. To align the coordinate system with the current viewpoint, enter **v**:

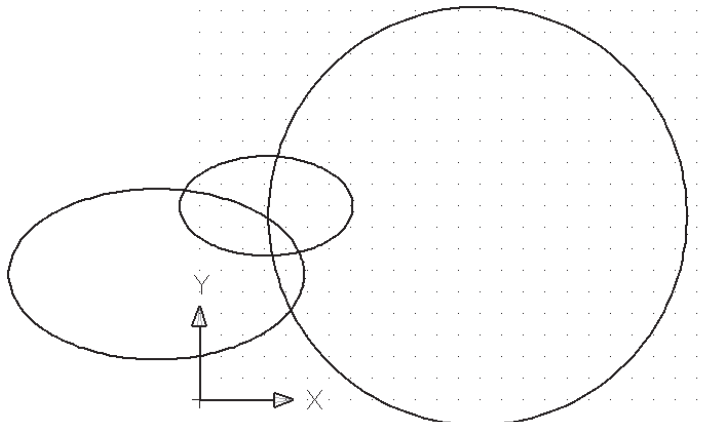
Specify origin of new UCS or [ZAxis/3point/ObJect/Face/View/X/Y/Z] <0,0,0>: **v**

This aligns the UCS to the current view, and is probably the easiest option — but also the least useful, because it does not necessarily align with features on 3D objects. (We'll see how to do this in the next tutorial.)



Notice that the UCS icon rotates to show just the x and y axes, just as if you were looking at the drawing in plan view. In addition, the grid “faces you.”

8. Draw another circle. Note that you are drawing it in the new x,y-working plane.



9. You can return to the WCS by clicking the  Earth-button in the View tab's Coordinates panel.

Alternatively, you can use the **UCS** command's **World** option:

Command: **ucs**

Enter an option [New/Move/orthoGraphic/Prev/Restore/Save/Del/Apply/?/World]
<World>: (Press **Enter**.)

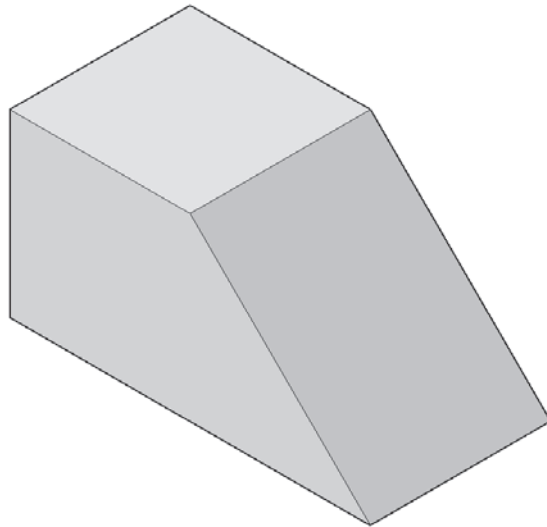
10. Return to plan view with the **PLAN** command.

By changing the orientation of the coordinate system to match the viewpoint of 3D objects, you can draw just as in plan view.

TUTORIAL: USING DYNAMIC UCS

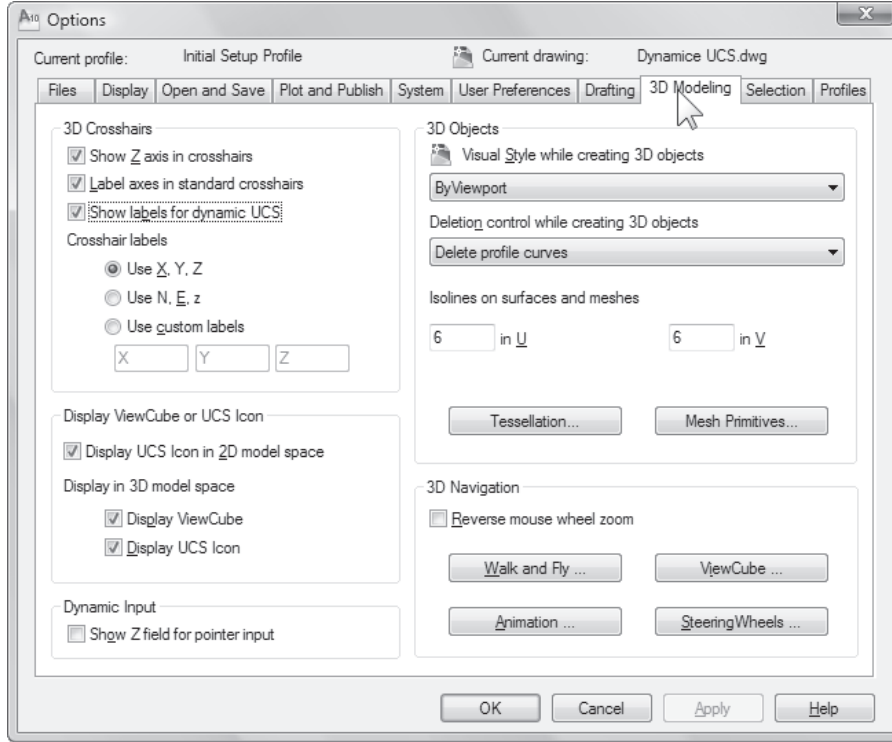
The UCS command is alright for setting up specific coordinate systems that you want to reuse frequently. In other cases, you just want the coordinate system to match the current working plane on-the-fly. This is where dynamic UCS is useful. When UCS is turned on, AutoCAD tries to match the UCS to the selected face, as described in this tutorial.

1. From the Companion CD, open the *Dynamic UCS.dwg* file, a drawing of a wedge-shaped structure. You will place text on three sides using dynamic UCS.

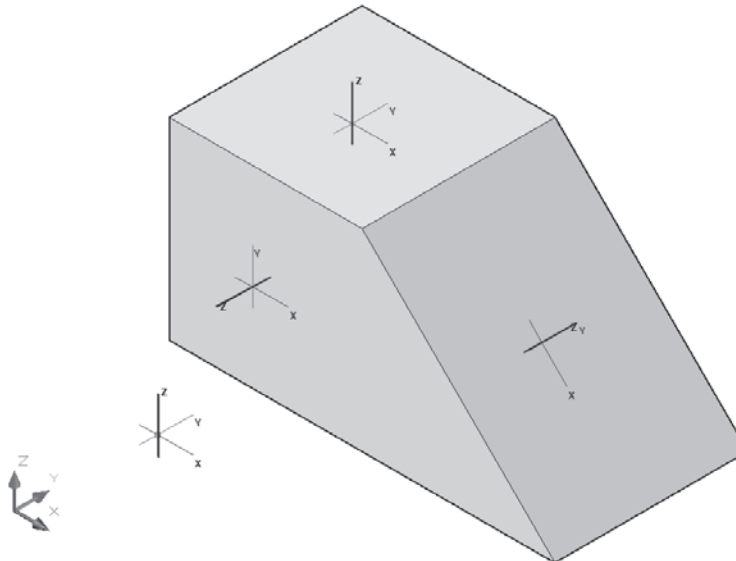


2. Ensure Dynamic UCS is turned on: its  button on the status bar should glow blue.
3. When working with dynamic UCSes, it is helpful to turn on cursor labels. These add X, Y, and Z to the axes of the 3D cursor:
 - a. Click the red A, and then click the **Options** button.
 - b. In the Options dialog box, click the **3D Modeling** tab.

- c. In the 3D Crosshairs section, turn on the **Show Labels for Dynamic UCS** option.



- d. Ensure that **Use X,Y,Z** is also turned on.
- e. Click **OK** to exit the dialog box.
4. To see dynamic UCS in action, start the **TEXT** command. At the 'Specify start point of text or [Justify/Style]:' prompt, move the cursor over the faces of the wedge. Notice that the cursor changes its orientation, depending on the angle of the face. I have superimposed four cursors in the figure below.



In all cases, the x, y axes of the cursor reorient so that they lie in the plane of the face. Similarly, the z axis always points away from the face. (I highlighted the z axis in the figure so that you can more easily see the change in orientation.)

5. Draw text on each of the three faces:

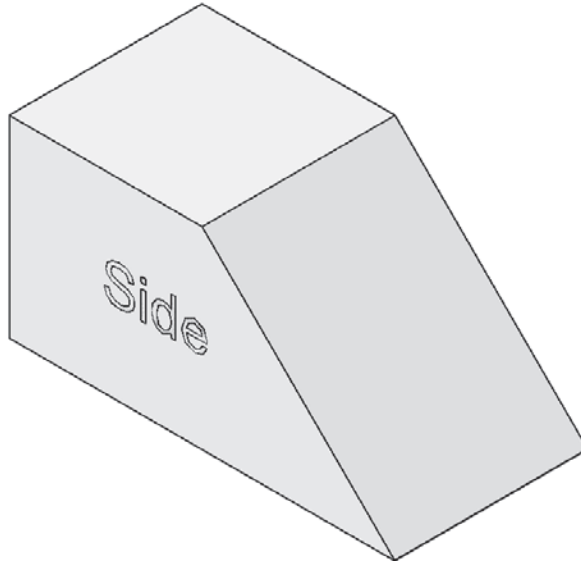
Command: **text**

Specify start point of text or [Justify/Style]: *(Pick a point on the side face.)*

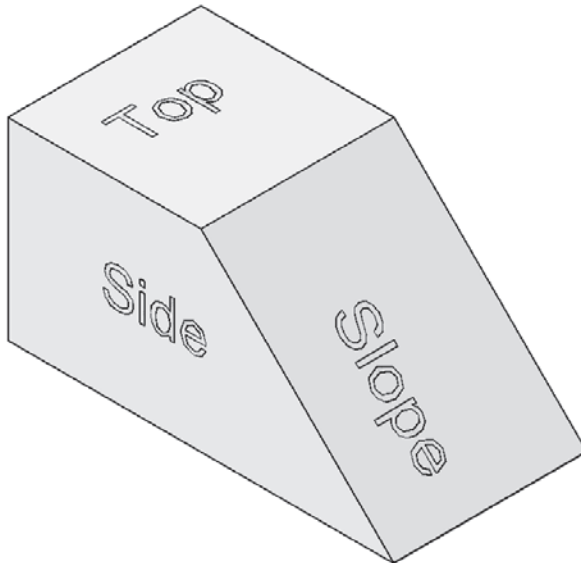
Specify height <0.2000>: *(Press **Enter**.)*

Specify rotation angle of text <0>: *(Press **Enter**.)*

Side *(Press **Enter**.)*



6. Repeat the **TEXT** command for the top and sloping faces. The result will look similar to the figure below.



Dynamic UCS is wonderful for letting you draw on any orientation of faces. Just remember, however, that it only kicks in during commands; it does not operate at the 'Command:' prompt. As well, dynamic UCS does not operate on curved faces, like those on spheres and cones; flat faces only!

JOINING SOLID MODELS

Sometimes you need to join two or more solid models. This is accomplished with the **UNION** command, and the result is known as a *body*.

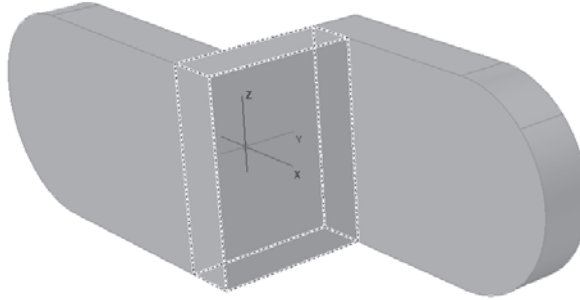
This command joins 2D regions into a single region, and 3D solid objects into a single solid model; regions and 3D solid models cannot, however, be mixed. The objects do not need to be touching or intersecting to be unioned.

To execute this command, AutoCAD needs to know just one thing from you: the objects to be joined.

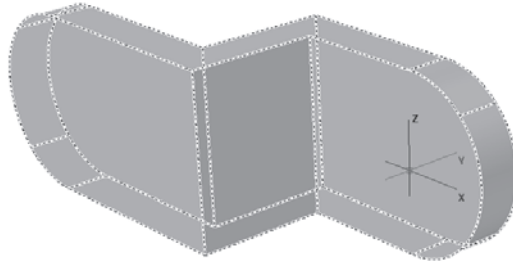
TUTORIAL: UNIONING SOLIDS

In this tutorial, you join three solid models into a single body.

1. From the Companion CD, open the *Union.dwg* file, a drawing of three solid models.
2. Move the cursor over the parts to confirm they are independent of each other; AutoCAD highlights just the one part at a time.

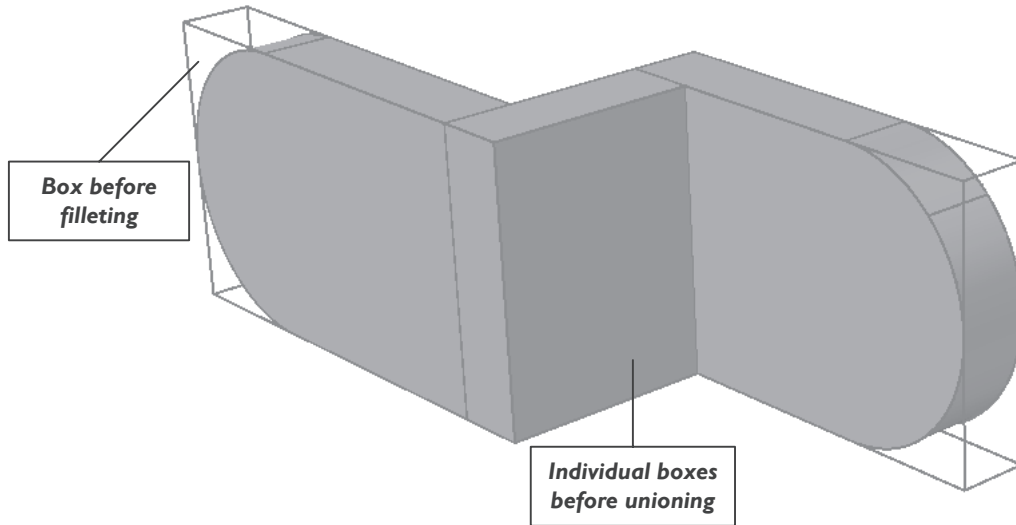


3. Join the three parts as follows:
 - a. In the Home tab's Solid Editing panel, choose **Union**.
 - b. In response to the prompt, enter **All** to select all objects in the drawing.
Select objects: **all**
Select objects: (Press **Enter**.)
 - c. Press **ENTER** to exit the command.
4. Move the cursor over the parts to confirm they are part of a single body; AutoCAD highlights all of them at once.

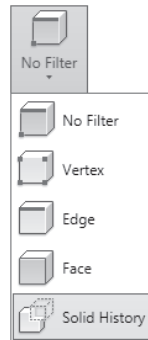


5. Even though the three are joined into one, you can still edit the originals with the **SHOWHIST** system variable. Change the value to **2**, as follows:
Command: **showhist**
Enter new value for SHOWHIST <1>: **2**

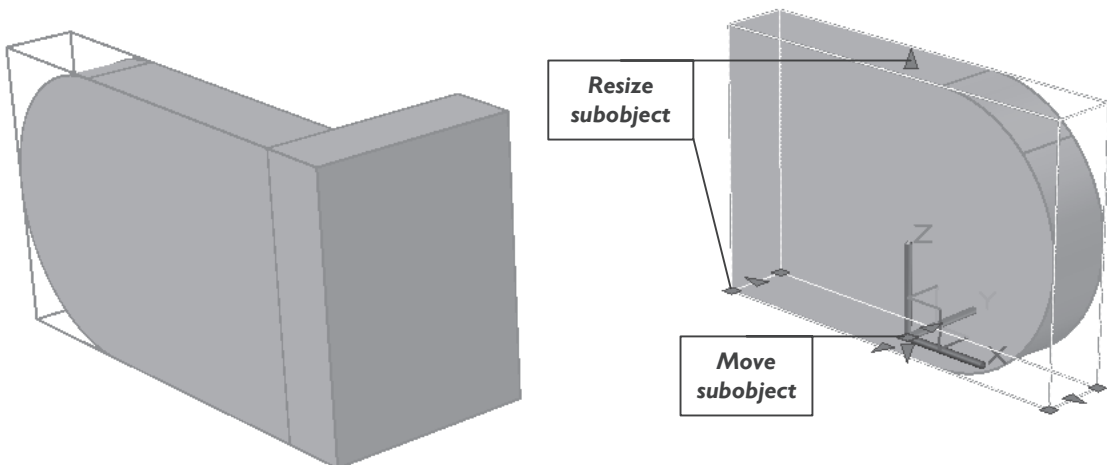
Notice that AutoCAD ghosts the outlines of the original solid models, including the boxes before their ends were filleted.



6. To edit the *subobjects* (individual parts of the solid body), follow these steps:
 - a. In the Home tab's Subobject panel, select **Solid History**.



- b. Hold down the **CTRL** key, and then select one of the subobjects.
The cursor shows the  icon, indicating it is in subobject selection mode.



Notice the grips; you can drag them to change the size of the subobject, even separate them by moving them away from each other.

- c. Press **ESC** to exit direct editing mode.
The subobjects still comprise a single solid body.

EXERCISES

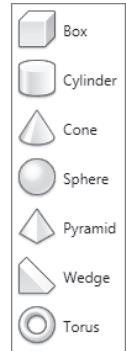
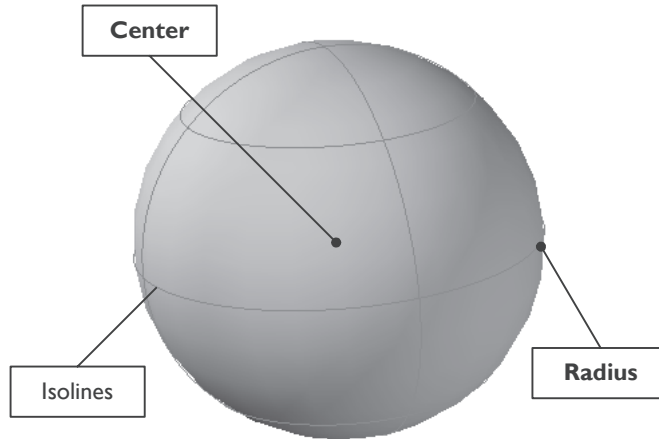
3D SOLID MODEL PRIMITIVES

Create 3D primitives with these modeling commands:

1. **SPHERE** command

Specify center point or [3P/2P/Ttr]: *(Pick a point.)*

Specify radius or [Diameter] <1.0>: **2**

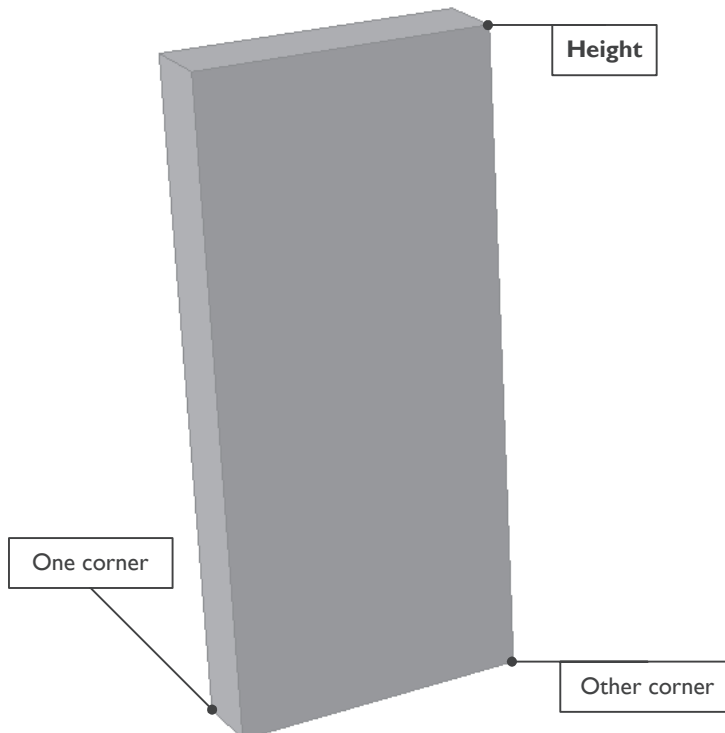


2. **BOX** command

Specify first corner or [Center]: *(Pick a point.)*

Specify other corner or [Cube/Length]: **1,4**

Specify height or [2Point] <2.0>: **9**

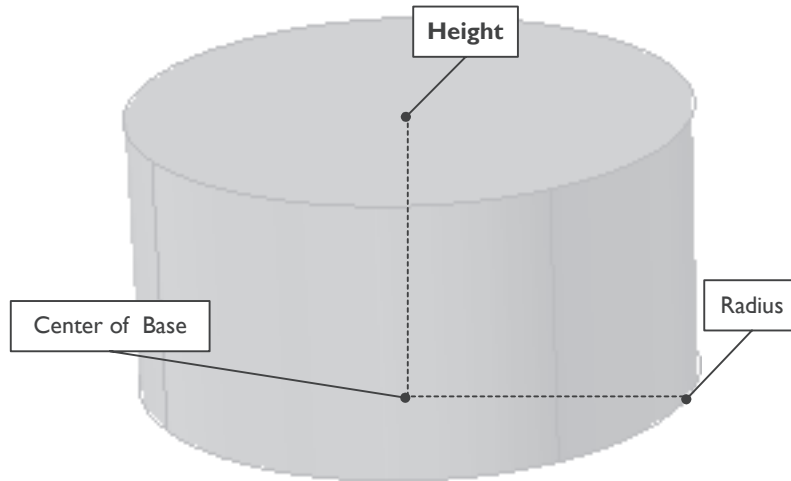


3. CYLINDER command

Specify center point of base or [3P/2P/Ttr/Elliptical]: *(Pick a point.)*

Specify base radius or [Diameter] <2.0000>: 2

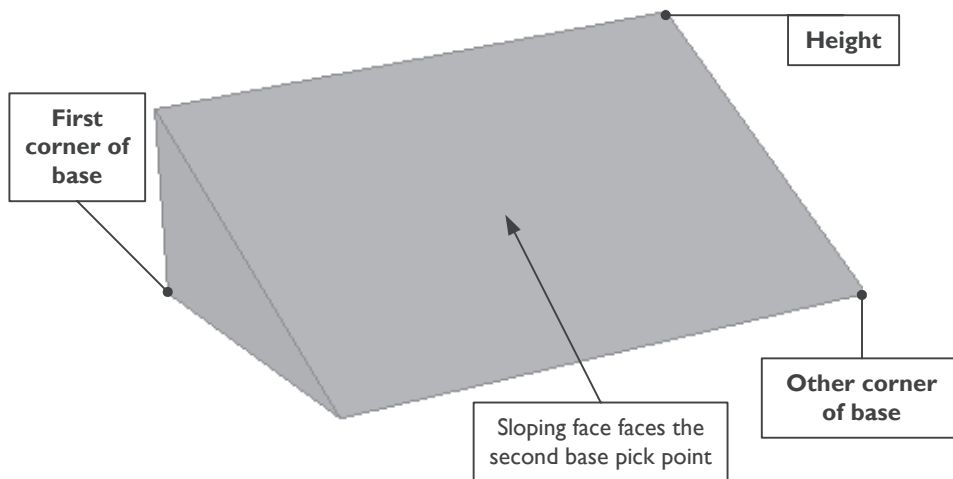
Specify height or [2Point/Axis endpoint] <9.0>: 2

**4. WEDGE command**

Specify first corner or [Center]: *(Pick a point.)*

Specify other corner or [Cube/Length]: @4,6

Specify height or [2Point] <2.0>: 2

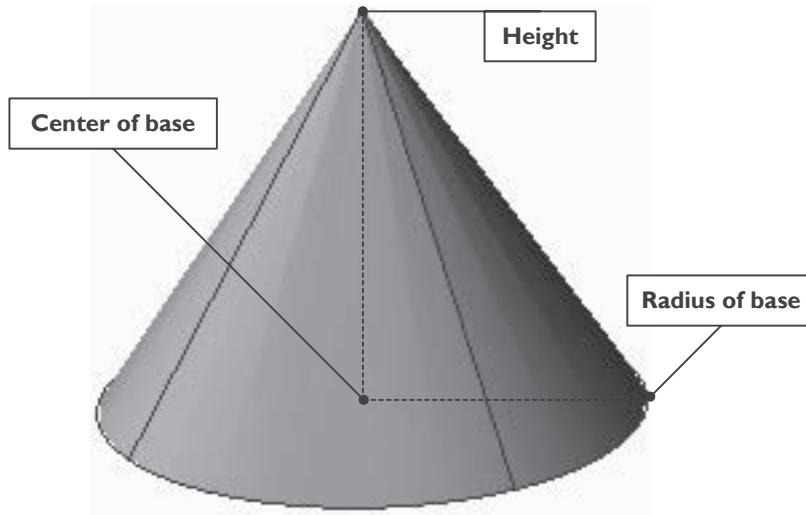


5. **CONE** command

Specify center point of base or [3P/2P/Ttr/Elliptical]: *(Pick a point.)*

Specify base radius or [Diameter] <2.0000>: 2

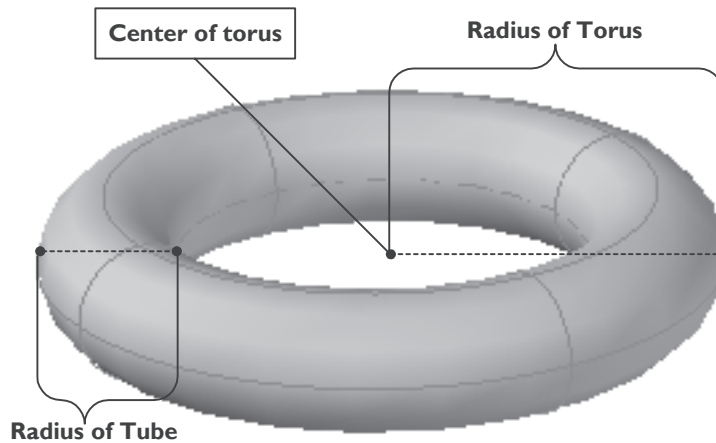
Specify height or [2Point/Axis endpoint/Top radius] <2.0000>: 3

6. **TORUS** command

Specify center point or [3P/2P/Ttr]: *(Pick a point.)*

Specify radius or [Diameter] <2.0000>: 2

Specify tube radius or [2Point/Diameter]: .5

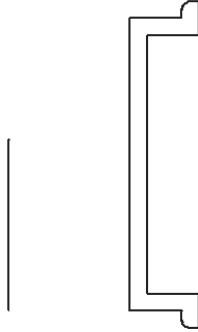


OTHER SOLIDS OPERATIONS

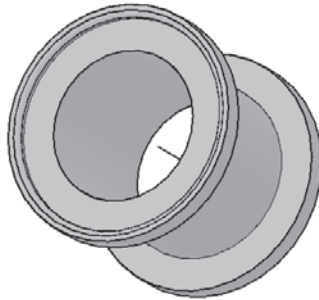
Construct 3D models from 2D objects using these modeling commands:

7. REVOLVE command

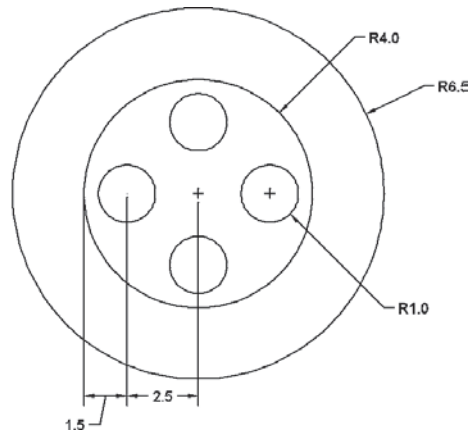
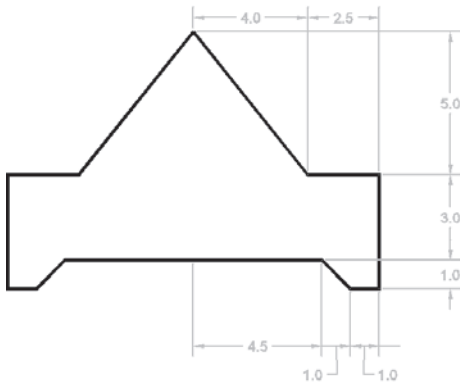
- a. From the Companion CD, open the *Revolve Exercise.dwg* file, a drawing of a spool profile.



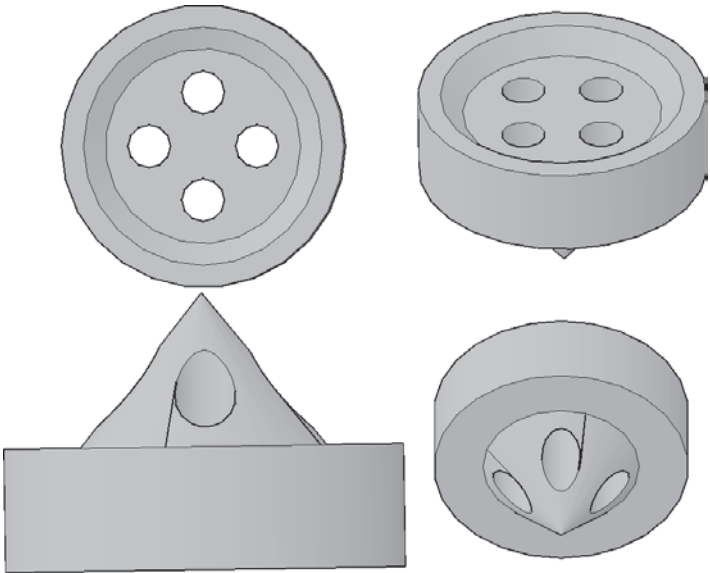
- b. Use the **REVOLVE** command to create the spool.
- c. Change the visual style to **Conceptual**.
- d. Drag the ViewCube to rotate the view to see the spool from another angle.



8. Create a 3D model of the flow reducer using these dimensions:

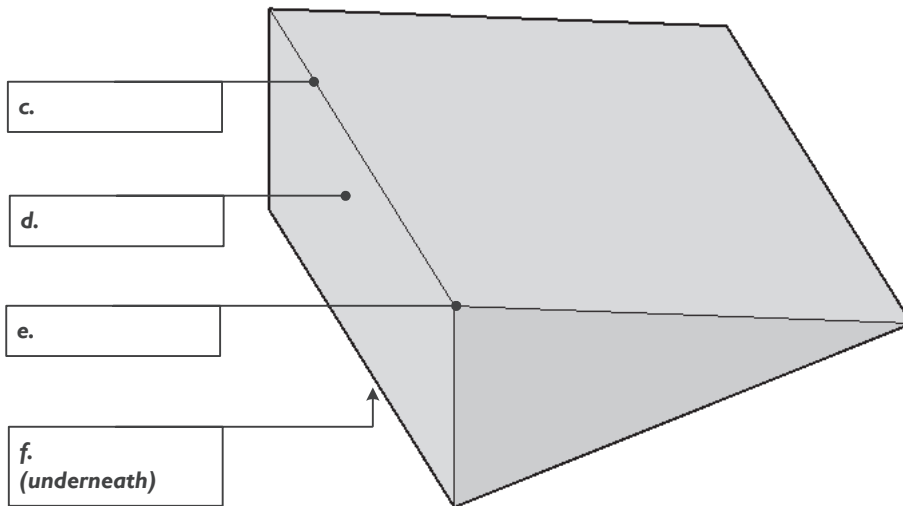
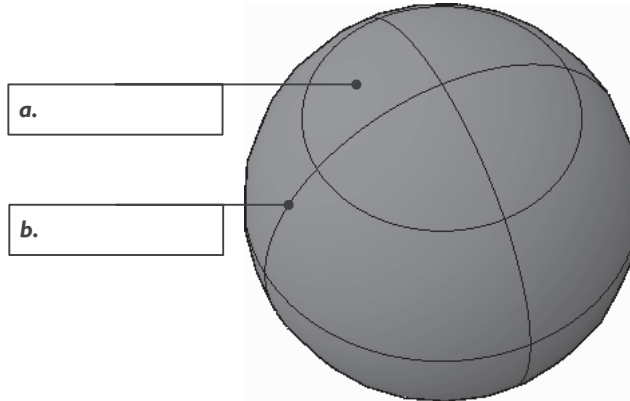


The result should look like the following views of the 3D model:



CHAPTER REVIEW

1. Where is most “3D drafting” performed in AutoCAD?
2. Are faces only flat?
3. Label the parts of the solid models illustrated below:



4. How do isolines affect the look of curved surfaces?
5. What are *subobjects*?
6. How do you select a subobject?
7. How does AutoCAD enable drafters to easily work on x,y-planes in 3D space?
8. Does dynamic UCS work on curved surfaces?
9. Describe the advantage of working in *isometric mode*.
10. What is the meaning of the following abbreviations?
 - a. WCS
 - b. UCS
11. Describe three functions performed by the UCS icon:
 - a.
 - b.
 - c.

12. How is the ViewCube useful?
13. What is a *primitive*?
14. Describe two ways to begin creating 3D models:
 - a.
 - b.
15. How are holes made in 3D models?
16. Which command joins two or more solid models?
Must the models be touching to be joined?
17. What is the name for a single solid model made up of two or more other solids?
18. Can bodies be edited?
If so, how?
19. Identify the following primitives:





Introduction to 3D Mesh Modeling

In the last chapter, you experienced the precision of 3D solid modeling; in this chapter, you learn about a second way of producing 3D models, through mesh modeling (new to AutoCAD 2010).

Three-dimensional mesh modeling is better than solid modeling for organic flowing shapes, such as for hair dryers and cell phones. Mesh modeling is especially popular for styling automobiles and consumer products.

To distinguish them from pre-2010 3D mesh objects made of polyfaces, Autodesk calls the new ones “full featured mesh objects” and “subdivision” meshes; in this book, they are named “3D mesh objects.”

In this chapter, you learn the following commands for constructing mesh models:

MESH creates 3D mesh primitives.

MESHOPTIONS specifies the number of tessellation lines for mesh primitives.

MESHSMOOTHMORE and **MESHSMOOTHLESS** increase and decrease the smoothness levels of 3D mesh models.

MESHCREASE and **MESHUNCREASE** add and remove creases from smoothed mesh models.

NEW TO AUTOCAD 2010 IN THIS CHAPTER

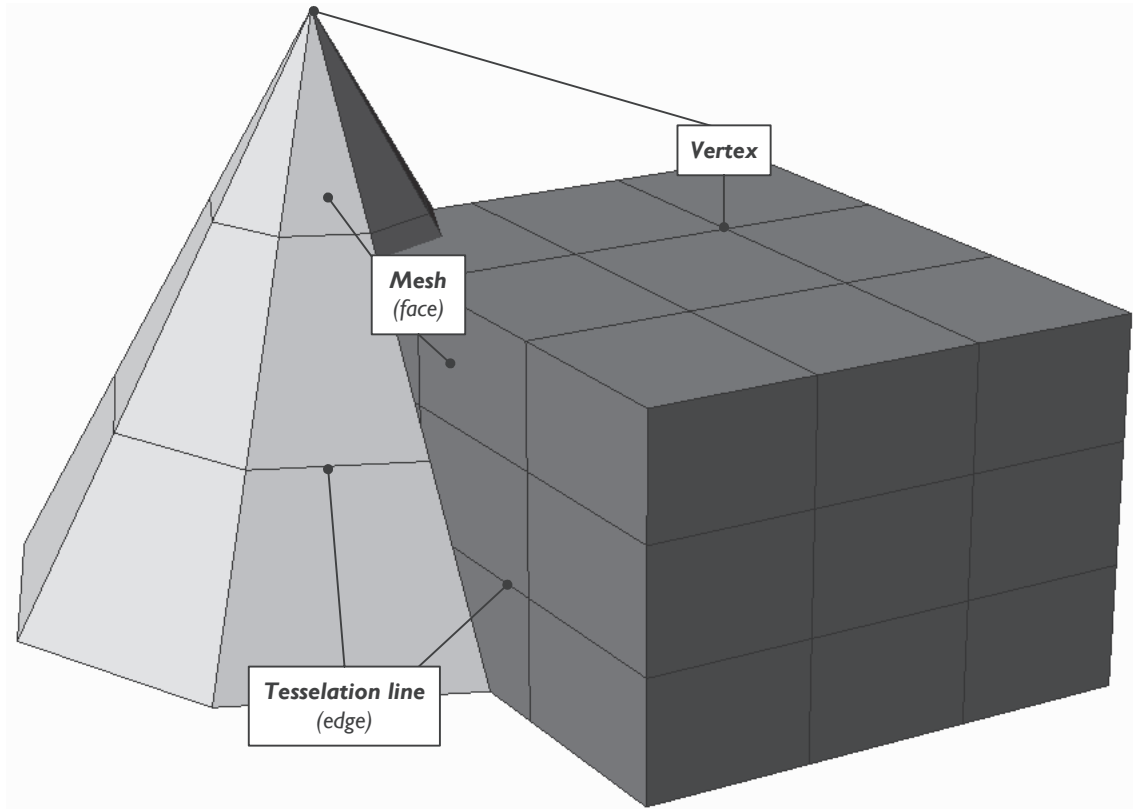
- All of the commands listed above are new to AutoCAD 2010.

PARTS OF 3D MESH MODELS

Mesh models differ from solid models in that they are hollow; you cannot perform analyses on mesh models, such as finding their mass or centroids.

Visually, 3D mesh models are covered by tessellation lines, whereas solid models sport lines only along their edge and isolines. Tessellation lines delineate the meshes that cover the surface; each face has one or more meshes.

Three-dimensional mesh models use much of the same vocabulary as solid models, as illustrated below. The primary difference is the use of tessellation lines in place of solid models' isolines.



Meshes cover the surfaces of mesh models. Generally, each mesh has four edges, although meshes on the bases of cylinders and cones are triangular. (In subobject selection, AutoCAD considers each mesh to be a face.)

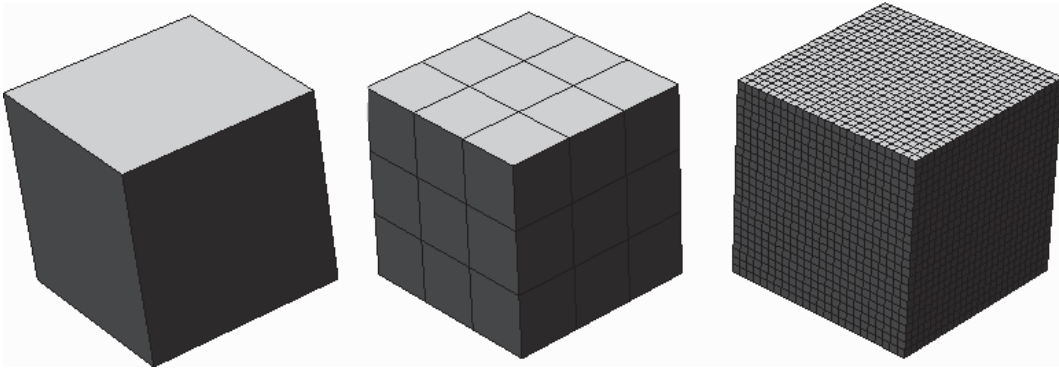
Tessellation lines define the edges of flat and curved meshes. (In subobject selection, AutoCAD considers each tessellation line to be an edge.)

Vertices are at the intersections of tessellation lines.

Bases are the bottoms of 3D models. When creating primitives like boxes and cylinders, you first define the size of the base and then the height. The bases of boxes are squares or rectangles; the bases of cones are circles or ellipses.

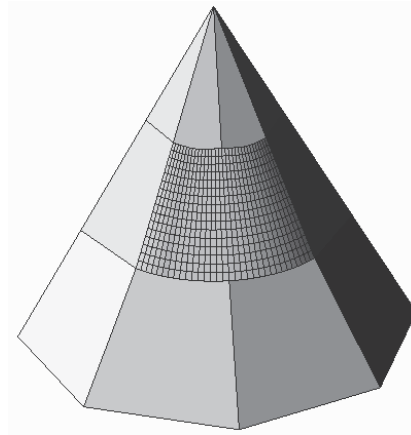
MULTIPLYING MESHES

The number of meshes on mesh models is whatever number you require for modeling. When first drawing a mesh model primitive, such as box or sphere, you specify the number of tessellation lines (which define meshes) with the MESHPRIMITIVEOPTIONS command.



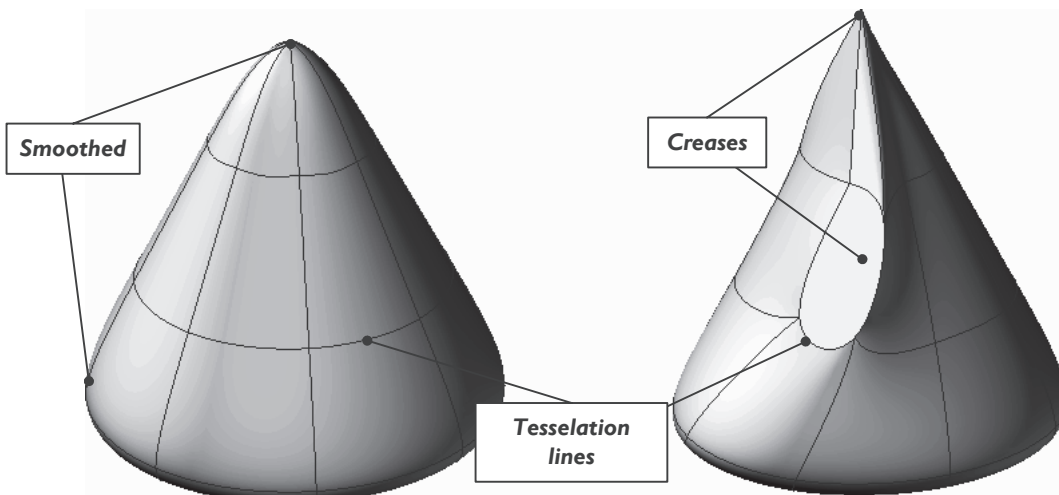
Left to right:
 Mesh density = 1 (minimum), 3 (default), and 24.

Later, if you find you need more meshes in one part of the model, you subdivide meshes with more meshes with the **MESHREFINE** command, as illustrated below. You increase mesh density selectively, because high mesh density slows down AutoCAD.

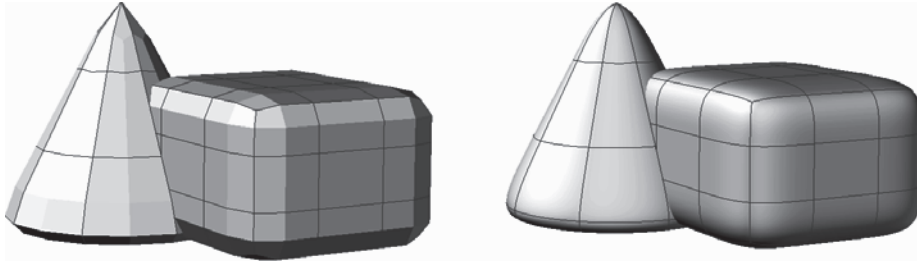


Increasing Smoothness

Mesh models are big on smoothness, a term appearing frequently in this chapter. Indeed, the entire point to meshes is that they can be rounded off (smoothed) to create those organic shapes. After mesh models are smoothed (rounded) with the **MESHSMOOTHMORE** command, you can selectively sharpen edges by *creasing* them with the **MESHCREASE** command.



Smoothing can be applied more than once, up to a maximum level of 4. Each application progressively rounds off the corners to the point that a cube looks nearly like a sphere. (Actually, the level can go higher than four, but your computer might not have the processing power required, subsequently slowing down operations. The level is set by SMOOTHESHMAXLEV system variable, to a maximum of 255.)



Left: Pyramid and box with smoothness level = 1.

Right: Smoothness level = 4.

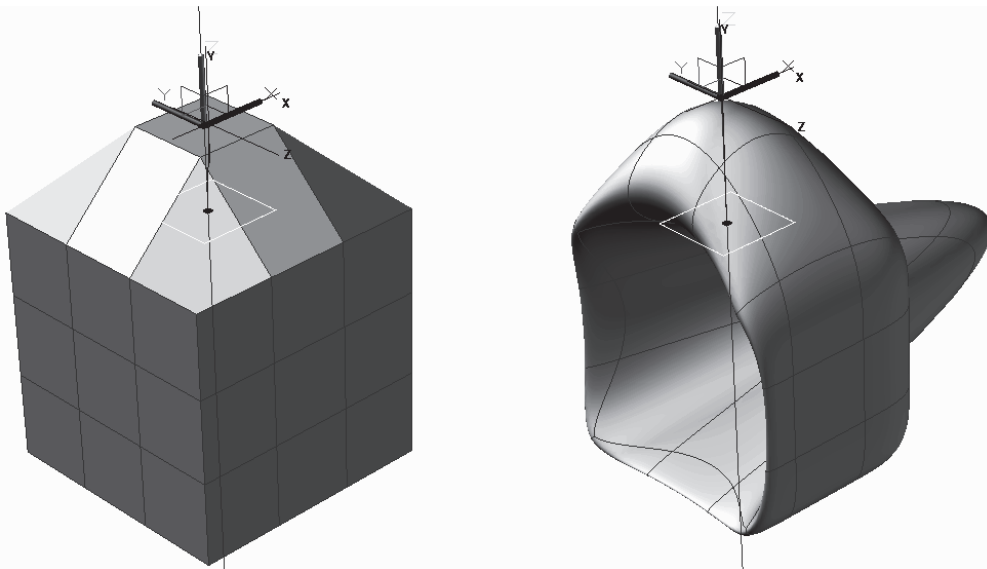
You can “un-smooth” smoothed mesh models with the MESHSMOOTHLESS command all the way back to level 0 where the hard edges reappear.

AutoCAD uses the term “smoothness” for two different operations, unfortunately. In one operation, the MESHSMOOTHMORE command smooths hard-edged mesh models; in a different operation, the MESHSMOOTH command converts 3D solid models and surfaces into mesh models, even though they won’t look smooth after conversion.

Manipulating Mesh Models

Mesh models were meant to be manipulated directly. You just grab a face, vertex, or edge, and then drag — no command needed. (You use the same subobject selection method as with 3D solid models.)

As you drag, adjacent faces drag along, as illustrated below. To create the organic shape shown at right, I merely maximized the smoothness (to level 4) of the cube, and then dragged two meshes — one on the left face and another on the top face. Total modeling time: under ten seconds.



Left: Dragging a face on an unsmoothed cube.

Right: Dragging faces on a smoothed cube.

Use the 3D gizmos to move, rotate, and scale faces in the x, y, or z directions. In the figures above, I selected the z axis (blue rod) of the 3D Move gizmo to ensure my dragging action was made precisely vertically.

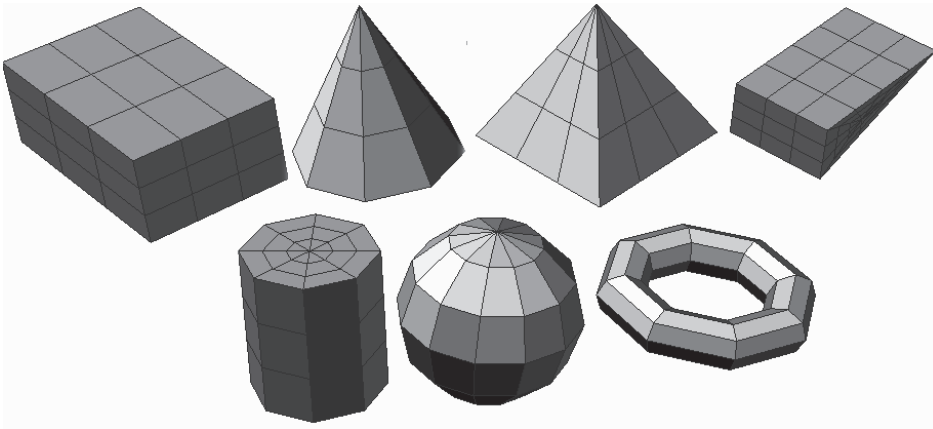
Creating and Editing 3D Mesh Models

3D models can be created in several ways: from primitives, as described next; from other objects, such as solid models and surfaces; and from mesh creation commands, like **TABSURF** and **LOFT**. Some of these are described in this chapter.

In addition to editing mesh models by smoothing, creasing, and dragging subobjects, you can edit them using Boolean operations, filleting and chamfering corners, and so on. The catch is that these operation cannot be applied directly to mesh models; they must first be converted to solid models.

TUTORIAL: CREATING 3D MESH PRIMITIVES (MESH)

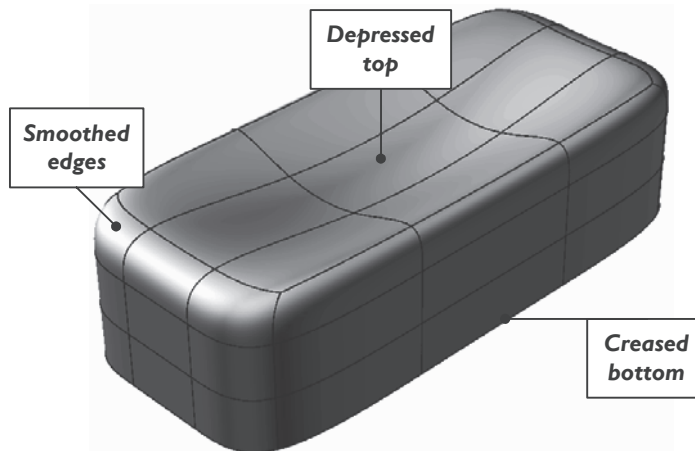
The **MESH** command creates 3D mesh primitives: square and rectangular boxes, pointy and flat top cones, round and elliptical cylinders, multisided pyramids, spheres, wedges, and tori.



Top row, left to right: Box, Cone, Pyramid, Wedge.

Bottom row: Cylinder, Sphere, Torus.

In this tutorial, you create a shape that looks like the keycap of a computer keyboard. Starting with a box, you smooth it, depress the top, and then crease the bottom edges. The finished 3D model will look something like the figure illustrated below.



1. Start a new drawing with AutoCAD using the *acad3d.dwt* template drawing.
2. Turn off the grid by clicking the **Grid** button on the status bar. Ensure the workspace is set to "3D Modeling."
3. Drawing meshes is easier with the ribbon than entering commands at the keyboard. To do so, click the **Mesh Modeling** tab in the ribbon.

- To draw the box, choose **Mesh Box** in the Primitives panel of the Mesh Modeling tab.



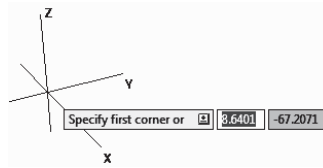
When you click the button, AutoCAD starts the command and enters the **B** (box) option on your behalf:

Current smoothness level is set to : 0

Enter an option [Box/Cone/CYLinder/Pyramid/Sphere/Wedge/Torus/SEttings] <Box>: B

- At the prompt, pick a point anywhere in the drawing:

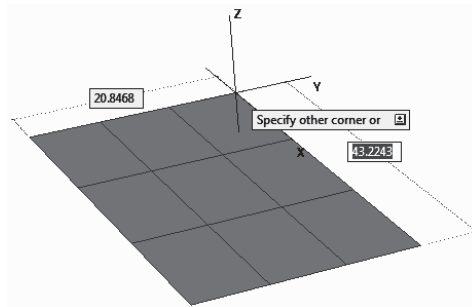
Specify first corner or [Center]: *(Pick a point to locate the first corner of the box.)*



Note that the current smoothness level = 0; this means the box will have sharp edges.

- Pick a second point to define the size of the base:

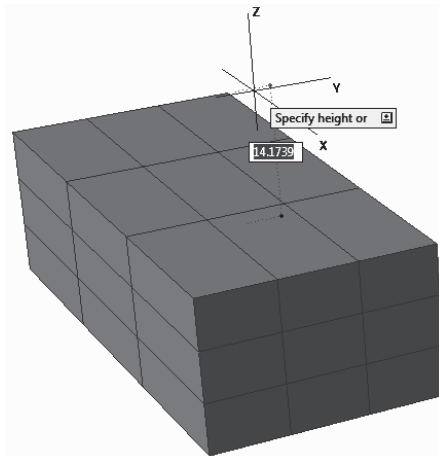
Specify other corner or [Cube/Length]: *(Pick another point to specify the area of the base.)*



Notice that the base is made of a 3x3 grid of meshes.

- Drag the cursor to indicate the height:

Specify height or [2Point]: *(Pick a point to specify the height.)*



The box is complete. In the next tutorial, you apply smoothing to the box.

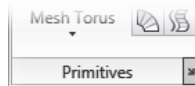
Setting Tessellation Lines

The MESH command uses a default number of tessellation lines, which define the number of mesh “rectangles” on the surface of mesh models. AutoCAD provides no fewer than three ways to change default values.

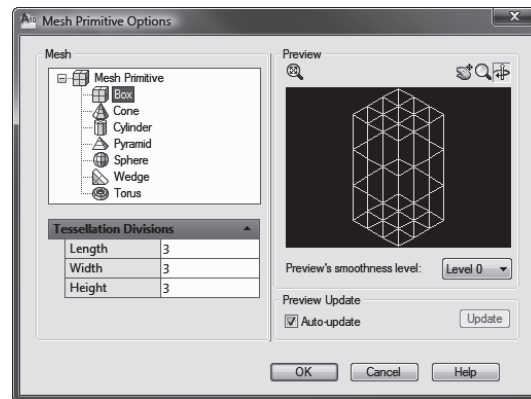
Here we look at two methods; the third method uses system variables and is much too cumbersome.

MeshOptions

The MESHOPTIONS command presets the number of tessellation lines for all primitives through a dialog box. To access the dialog box, click the diagonal arrow on the Primitives panel, as illustrated below.

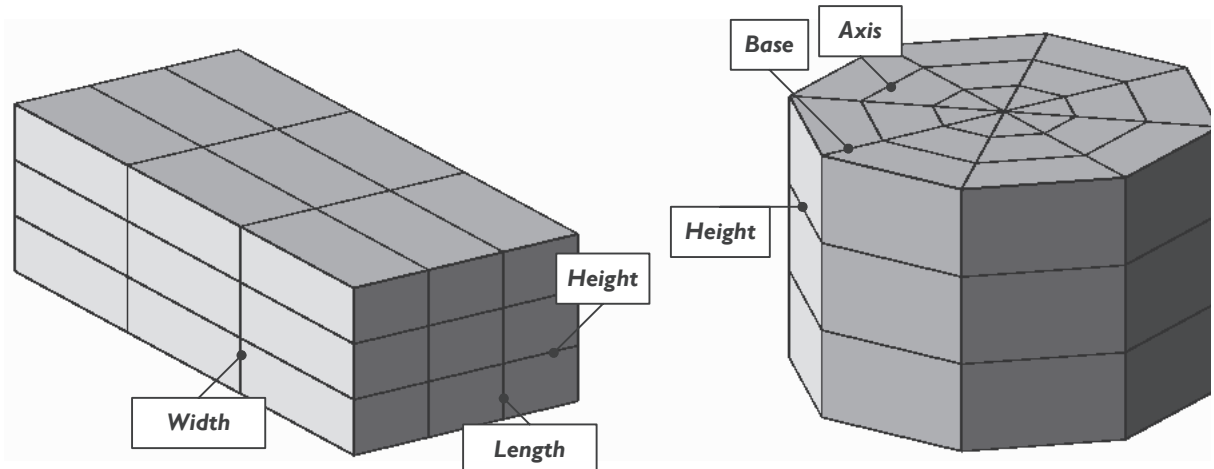


The Mesh Primitive Options dialog box previews each of the primitives as you change the number of tessellation lines.



The minimum number is usually 1, the maximum 256:

- **Length** specifies the tessellation lines along the length of boxes (x axis); default = 3.
- **Width** specifies the tessellation lines along the width of boxes (y axis).
- **Height** specifies the tessellation lines along the height of boxes (z axis).



For cones, the divisions are specified by different names:

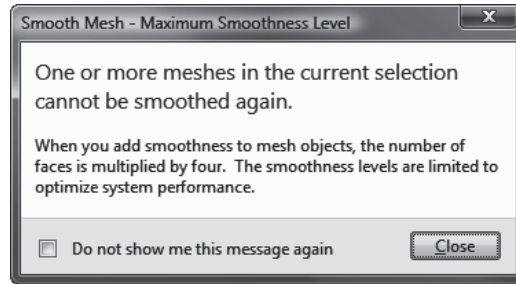
- **Axis** specifies the tessellation lines around the perimeter of the cylinder's base. The minimum value is 3, which draws a cylinder with a triangular base; the same restriction applies to the axes of cones and spheres, as well as to all parameters of tori.

- **Base** specifies the tessellation lines between the perimeter and center of the cylinder's base.
- **Height** specifies the tessellation lines between the base and top of the cylinder.

Remember to avoid entering large values for Tessellation Division, because they slow down AutoCAD.

MESHSMOOTHMORE AND MESHSMOOTHLESS

The MESHSMOOTHMORE command rounds off the corners of 3D mesh models. Each time you apply the command, the model is smoothed some more — up to the maximum level set by SMOOTHMESHMAXLEV, which is set to 4 by default. When you go beyond the limit, AutoCAD warns you with this dialog box:



The complementary command is MESHSMOOTHLESS, which reduces the level of smoothness by one step until it reaches 0.



Note You can draw mesh primitives with any level of smoothness with the **MESH** command's **SEttings** option, as follows:

Command: **mesh**

Current smoothness level is set to : 0

Enter an option [Box/Cone/CYLinder/Pyramid/Sphere/Wedge/Torus/SEttings] <Box>: **se**

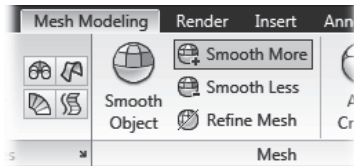
Specify level of smoothness or [Tessellation] <0>: *(Enter a value between 0 and 4.)*

The **Tessellation** option displays the Mesh Primitive Options dialog box described earlier.

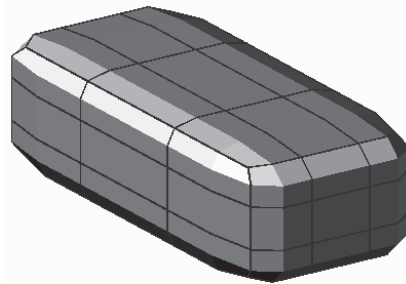
TUTORIAL: SMOOTHING MESH MODELS

In this tutorial, you apply smoothing to the box created in the earlier tutorial.

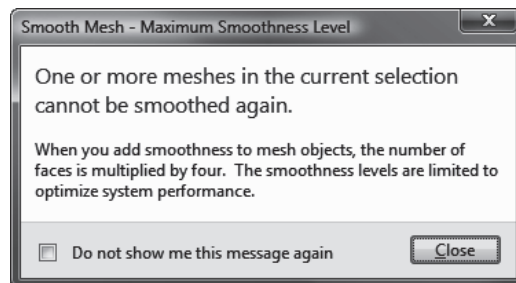
1. From the 3D Modeling ribbon's Mesh Modeling tab, choose the **Smooth More** button in the Mesh panel. (Don't click Smooth Object, which has a different function!)



2. Pick the box. Notice that its edges are somewhat rounded off (smoother).
Select mesh objects to increase the smoothness level: *(Pick the box.)*
Select mesh objects to increase the smoothness level: *(Press **Enter** to exit the command.)*

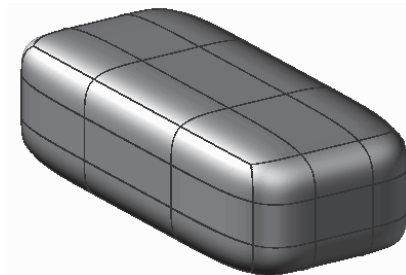


3. Press **SPACEBAR** to repeat the command, and then choose the box again.
Repeat until AutoCAD presents its warning dialog box.



Note The dialog box indicates the box has reached the maximum level of smoothness (4). If you have a really powerful computer, you can increase the limit to 255 with the **SMOOTHMESHMAXLEV** system variable — although Autodesk recommends using 5 as the maximum.

5. Click **OK** to exit the dialog box. At this point, the box should look something like the one illustrated below.



In the next tutorial, you depress the top of the box.

MOVING, ROTATING, AND SCALING MESH FACES, EDGES, AND VERTICES

There is no command for manipulating the surface of 3D mesh objects. You choose a face, edge, or vertex, and then drag. You can use 3D gizmos to move, rotate, and scale faces, as illustrated below.



Left: Moving a face (original highlighted in white).
Center: Rotating a face.
Right: Scaling a face larger (highlighted in white).

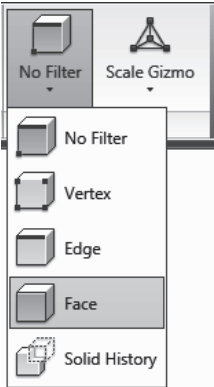
The dynamic UCS icon lets you choose an axis or plane along which to perform the transformation, such as along the z axis or in the x, y-plane.

TUTORIAL: MOVING FACES

In this tutorial, you move a face to create a depression in the top of the box.

1. Before manipulating a 3D mesh object, you need to decide on the subobject to be manipulated. For this tutorial, you will be moving a face, so you should set the subobject selection mode to Face.

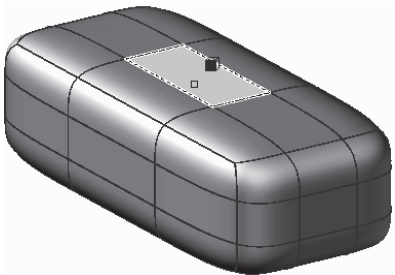
In the Subobject panel (Mesh Modeling tab), choose **Face**.



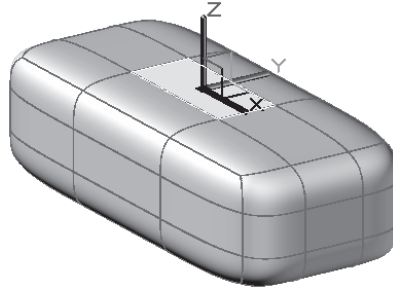
2. To choose a subobject, hold down the **CTRL** key. (If you do not, you end up selecting the entire object.) Notice the Face icon next to the square pickbox cursor: AutoCAD confirms it is in face subobject selection mode.



3. Move the cursor over the box. Notice that AutoCAD highlights individual faces.



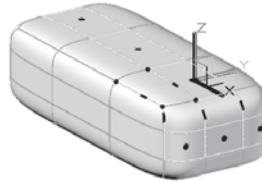
4. Click the center face on the top of the box. Notice the 3D Move gizmo (which looks like the UCS icon).



Somewhat harder to see is the red dot at the center of the face. This is AutoCAD's feedback indicating which face has been selected.

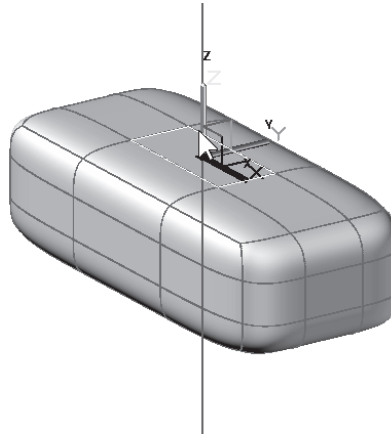


Note You can select more than one face for editing: click on each face while holding down with the **CTRL** key. As you choose additional faces, AutoCAD places a small red dot in each one.



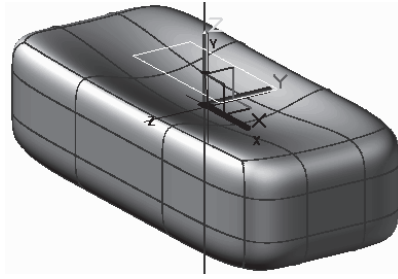
SHIFT+click removes subobjects from the selection set. (If you were choosing edges, AutoCAD marks them with short red lines; selected vertices are also marked with red dots. You can freely select faces, edges, and vertices by selecting the appropriate button in the Subobject panel.)

5. Move the cursor over the blue z axis of the 3D Move gizmo. Notice that it turns gold, and that a blue line shoots laser-like from each end of the axis. Click to choose the z axis.



Note If you want to rotate or scale the face, now is the time to switch modes. Right-click the z-axis, and then choose the mode from the shortcut menu.

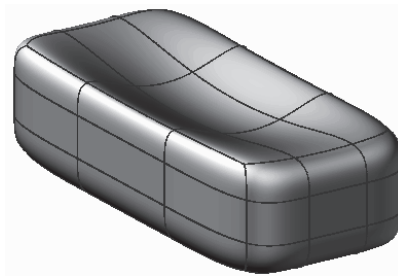
6. With the z axis selected, move the cursor. As you do, notice that the top of the box bulges out or inward, depending on the cursor movement.



Notice also that adjacent faces are also moved, but to a less extent. This ensures that the 3D mesh model remains smooth.

For this tutorial, move the cursor down (negative z direction) to create a mild depression. This is the top of the keycap.

7. Click to position the depression. Press **ESC** to exit face selection mode.



In the next tutorial, you sharpen the base of the box.

MESHCREASE AND MESHUNCREASE

You've seen how 3D mesh models are all about smoothness. But there are times you may want certain edges to be sharp — named a “crease” in this business. The **MESHCREASE** command applies creases to subobjects, not to entire mesh models.

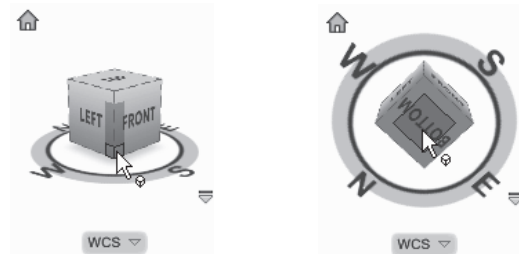
There is a tricky aspect to creasing. Specifically, crease values between 1 and 6 cause the crease to disappear when its value is less than the smoothness level of the mesh. Fortunately there is a solution to the headscratcher: enter **A** for Always, and the crease won't disappear.

The **MESHUNCREASE** command removes creases from subobjects.

TUTORIAL: ADDING CREASES

In this tutorial, you add creases to the base of the box.

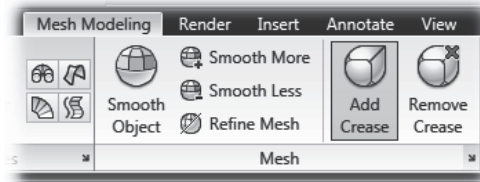
1. Rotate the model so that you can see the bottom more clearly. Drag the corner ViewCube until you see the word Bottom.



Left: Dragging the ViewCube by a corner.

Right: ViewCube rotated to show the bottom of the 3D model.

2. You will be creasing the edges (tessellation lines) of the bottom of the box, so change subobject selection to **Edge**.
3. In the Mesh panel, choose **Mesh Crease**.



4. AutoCAD prompts you to select subobjects. Pick an edge making up the base of the box. (No need to hold down the **CTRL** key).

Select mesh subobjects to crease: *(Choose an edge.)*

Notice that AutoCAD highlights selected edges in white. (No red dots or lines appear this time.)

5. Continue, picking one by one the remaining edges making up the base of the box. In total, you pick 12 edges.



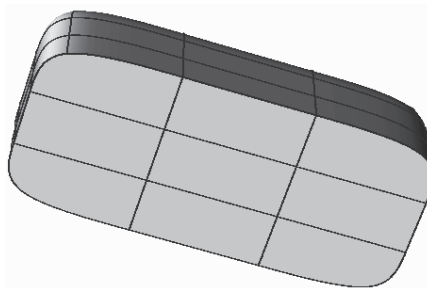
If you pick an incorrect edge, enter **u** to undo the last chosen point. (**Undo** is a hidden option in this command.)

6. When done, press **ENTER**.

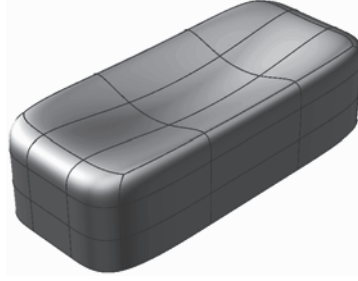
Select mesh subobjects to crease: *(Press **Enter**.)*

7. AutoCAD asks for the crease value. While the value can range from 0 to 6, enter **A** for “Always” to keep the crease there, no matter how the smoothness level of the rest of the model may change. (Entering **A** is the same as entering 0.)

Specify crease value [Always] <Always>: **a**



8. Use the ViewCube to rotate the view of the keycap.
(As a shortcut, click the tiny **house** icon near the ViewCube: this rotates the model back to the “home view,” which is the default SW Isometric viewpoint.)



The tutorial is complete. Save your work under the name of *Key cap.dwg*.

EXERCISES

3D SURFACE MODEL PRIMITIVES

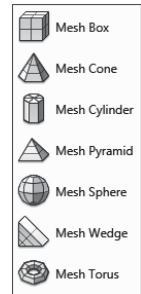
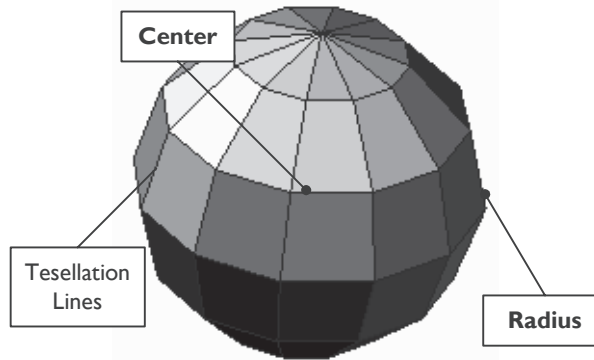
Create 3D primitives with these modeling commands:

1. **MESH** command's **Sphere** option:

Current smoothness level is set to : 0

Specify center point or [3P/2P/Ttr]: *(Pick a point anywhere in the drawing.)*

Specify radius or [Diameter] <1.0>: 2

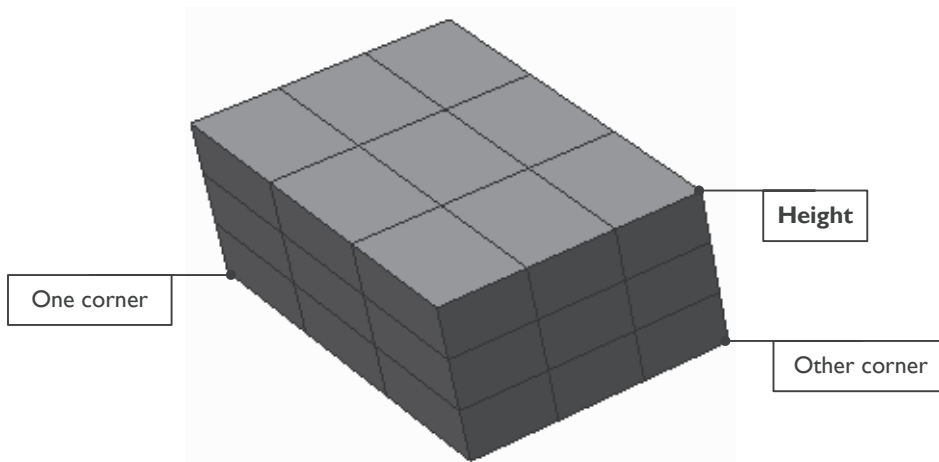


2. **MESH** command's **Box** option:

Specify first corner or [Center]: *(Pick a point anywhere in the drawing.)*

Specify other corner or [Cube/Length]: 3,4

Specify height or [2Point] <1.0>: 2

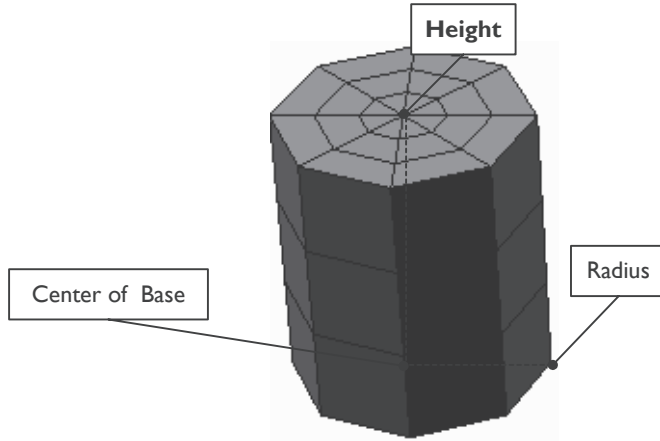


3. **MESH** command's **Cylinder** option:

Specify center point of base or [3P/2P/Ttr/Elliptical]: *(Pick a point anywhere in the drawing.)*

Specify base radius or [Diameter] <3>: 2

Specify height or [2Point/Axis endpoint] <1">: 2

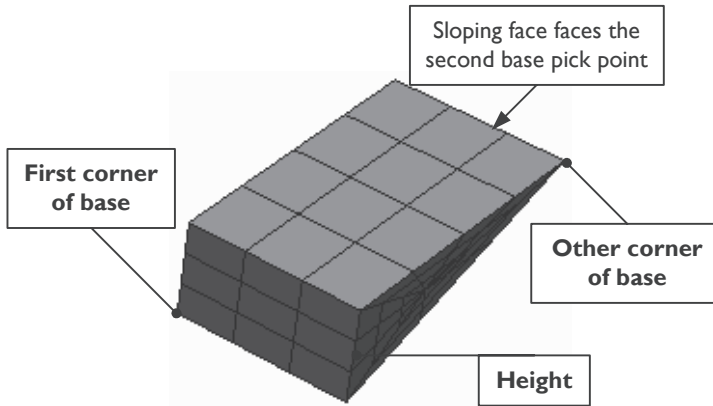


4. **MESH** command's **Wedge** option:

Specify first corner or [Center]: *(Pick a point anywhere in the drawing.)*

Specify other corner or [Cube/Length]: @4,6

Specify height or [2Point]: 2

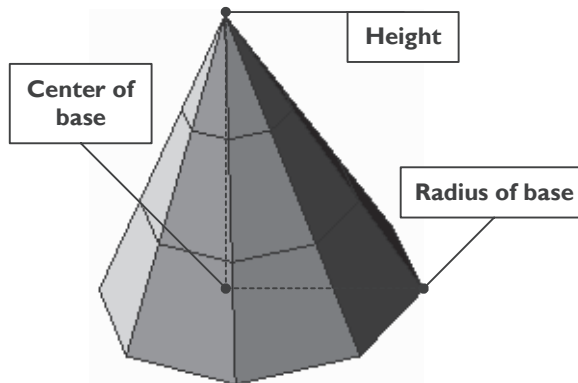


5. **MESH** command's **Cone** option:

Specify center point of base or [Edge/Sides]: *((Pick a point anywhere in the drawing.))*

Specify base radius or [Inscribed]: 3

Specify height or [2Point/Axis endpoint/Top radius]: 5

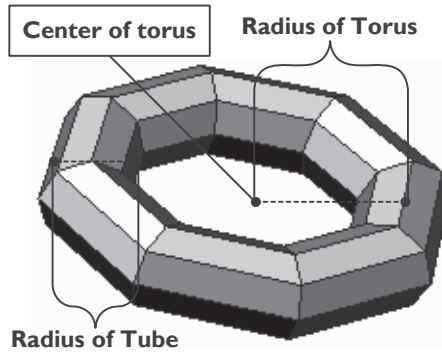


6. **MESH** command's **Torus** option:

Specify center point or [3P/2P/Ttr]: (Pick a point anywhere in the drawing.)

Specify radius or [Diameter] <1.0>: 4

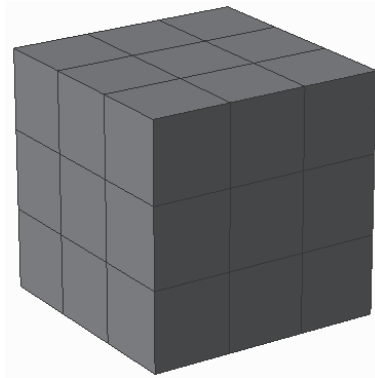
Specify tube radius or [2Point/Diameter]: 1



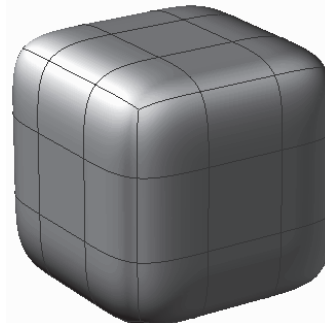
OTHER MESH OPERATIONS

Modify 3D surface models.

7. a. In a new drawing, draw a cube.
With which command did you draw the cube?



- b. Smooth the box.
Which command smooths 3D mesh models?
c. Repeat until the box is at its maximum level of smoothness.



- d. Use the **MESHSMOOTHLESS** command to unsmooth the box.

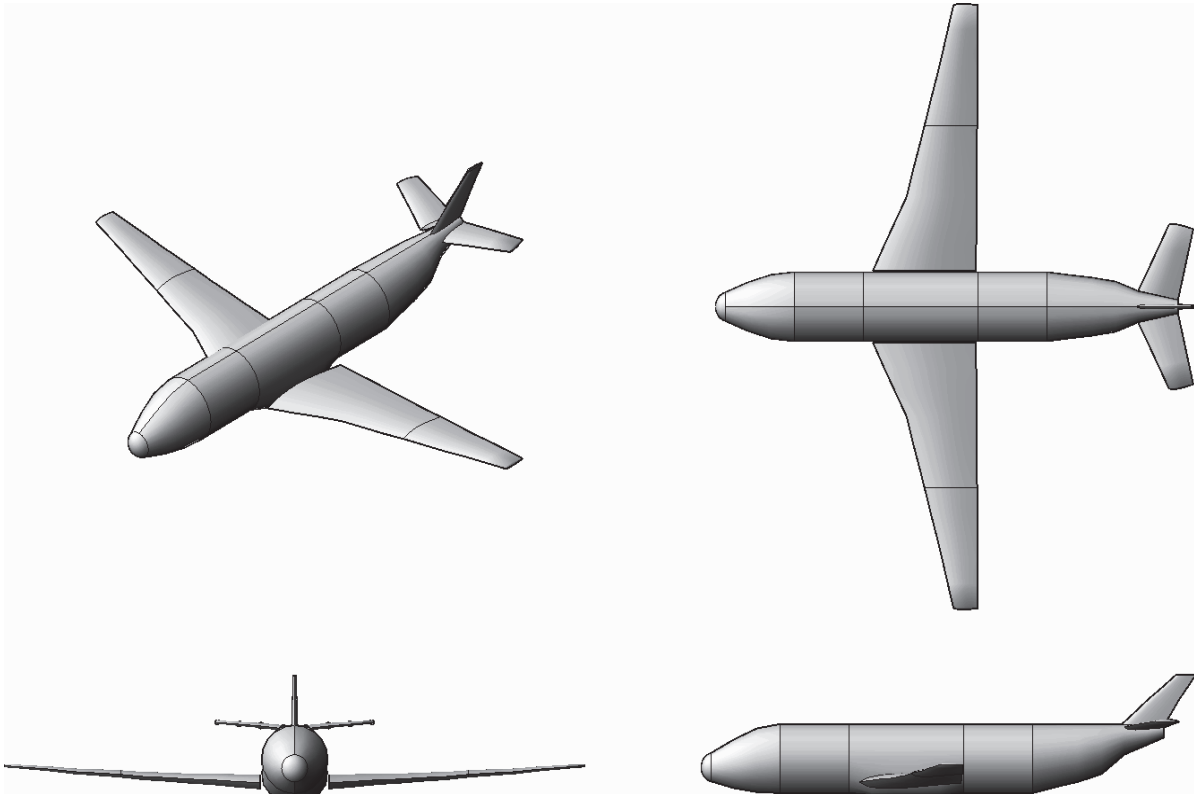
8. Create a 3D mesh model of the deflated basketball using these commands:
 - a. **MESH** command's **Sphere** option.
 - b. **MESHSMOOTHMORE** command.
 - c. Subobject selection set to **Face**.
 - d. Rotate face.

9. Create a 3D mesh model of the topless ballcap using these commands:
 - a. **MESH** command's **Torus** option.
 - b. **MESHSMOOTHMORE** command.
 - c. Subobject selection set to **Face**.
 - d. Scale face.

10. Recreate the apple as a 3D mesh model. It is approximately 2.5" (64mm) in diameter, and sits on a flat base.

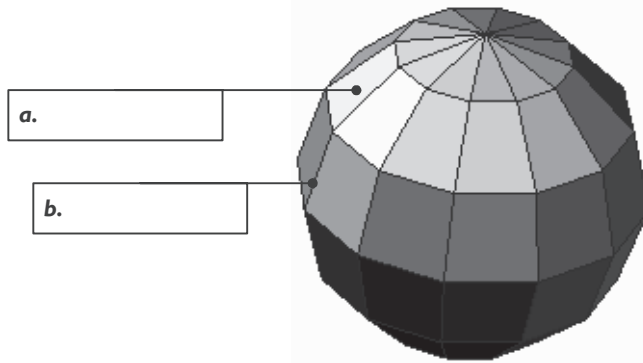


11. Create the airplane body as a 3D mesh model.



CHAPTER REVIEW

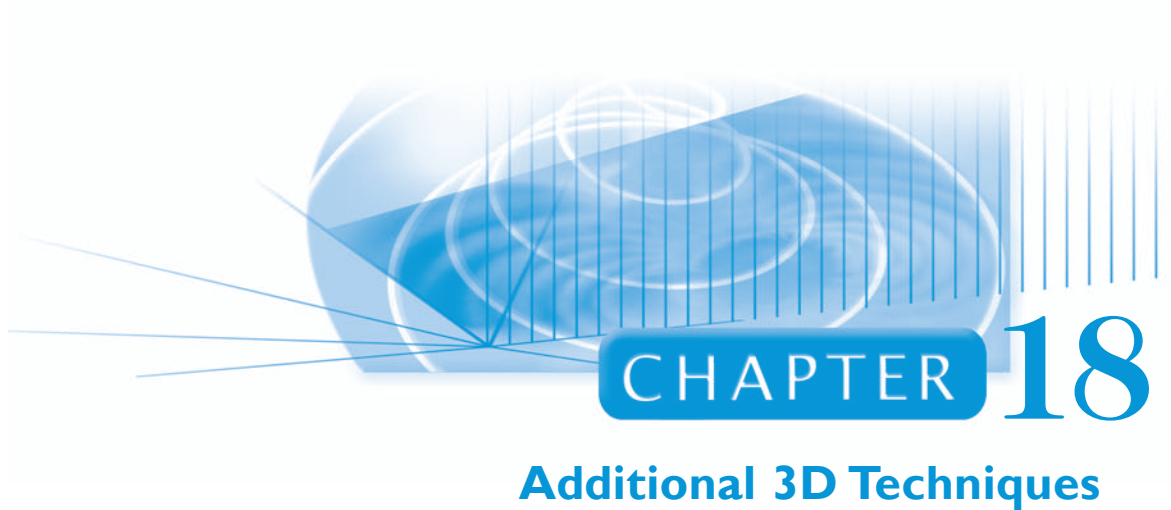
1. List two ways in which 3D mesh models differ from 3D solid models:
 - a.
 - b.
2. Label the parts of the mesh models illustrated below:



3. What is the purpose of *tessellation lines*?
4. Must meshes always have four straight sides?
5. Which command distorts meshes?
6. How do you select a subobject on a 3D mesh model?
7. What is a *crease*?
How do creases affect meshes?
8. What happens when mesh smoothness exceeds 4?
What happens when tessellation density has a large value?
9. Which command increases the smoothness of 3D mesh models?
10. What is a *primitive* in terms of 3D mesh models?
11. Can 3D mesh models be edited?
12. Identify the following primitives:

- a. 
- b. 
- c. 
- d. 
- e. 
- f. 
- g. 

13. What happens when you click the Home icon in the ViewCube useful?
14. Can more than one face or edge be selected for editing?
If so, how?
15. Why might you want to increase the number of meshes?



CHAPTER 18

Additional 3D Techniques

The previous two chapters introduced you to 3D modeling with solids and meshes. In this chapter, you learn how about other types of 3D modeling, such as creating surfaces and editing 3D models.

EDGESURF fits meshes between four connected edges.

RULESURF stretches meshes between two disconnected edges.

TABSURF raises meshes from path curves and direction vectors.

REVSURF revolves meshes from path curves and axes of rotation.

EXTRUDE and **SWEEP** create 3D objects from 2D designs.

UNION, **SUBTRACT**, and **INTERSECT** commands create new 3D bodies through Boolean operations.

INTERFERE creates 3D bodies from intersecting solids.

CHAMFER applies angular cutoffs to 3D models.

MASSPROP reports the properties of 3D solids.

NEW TO AUTOCAD 2010 IN THIS CHAPTER

- The **EDGESURF**, **RULESURF**, **TABSURF**, and **REVSURF** commands create now 3D mesh objects.
- The **UNION**, **SUBTRACT**, and **INTERSECT** commands now work with 3D mesh objects, after converting them to solid models.

2D EDITING COMMANDS THAT WORK IN 3D

Some 2D editing commands you learned about earlier in this book can be used on 3D objects. They work as you would expect. For instance, erasing and rotating 3D solid models are just like erasing and rotating 2D objects.

- **ALIGN** moves, resizes (scales), and/or rotates solid models in 3D space.
- **CHAMFER** cuts the edges of faces.
- **ERASE** erases the selected solid models from the drawing.
- **EXPLODE** converts 3D solid models into 2D regions, and 3D meshes into polyfaces.
- **FILLET** rounds the edges of faces.
- **MIRROR** and **MIRROR3D** make mirror copies of solid objects.
- **MOVE** moves solid models in 3D space (by any distance in the x, y, and/or z directions).
- **ROTATE** and **ROTATE3D** rotate solid objects in two and three dimensions.
- **SCALE** changes the size of solid objects.
- **CHANGE**, **CHPROP**, and **PROPERTIES** change only some properties of solid and mesh models.

2D COMMANDS THAT DON'T WORK

Not all 2D editing commands work with solid models. The **STRETCH** command only moves solid models, but does not stretch them. **EXTEND** and **LENGTHEN** apply only to open objects, which solid models are not; the workaround is to use the **SOLIDEDIT** command (not covered by this book.)

BHATCH, **HATCH**, and **HATCHEDIT** don't work, because solid and mesh models cannot be hatched. The closest workaround is to change their color with the **COLOR** command; alternatively, use the **MATERIAL** command to apply a material to the solid model, which is then displayed by the Realistic visual style.

You cannot break solid models with the **BREAK** command. As a workaround, use the **SLICE** command to cut off a portion of the solid model.

DIVIDE and **MEASURE** do not apply to solid models. The **TRIM** command does not work with solid models; the workaround is to use the **INTERSECT** or **SLICE** command. 3D models cannot be offset with the **OFFSET** command; the workaround is to use the **COPY** or **SOLIDEDIT** command.

Most grips editing options work with 3D solids: Copy, Rotate, Move, and Scale. The Stretch option moves them, while the Mirror option rotates.

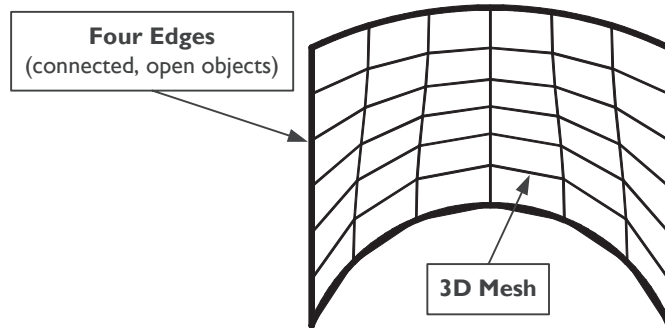
AutoCAD LT

AutoCAD LT can view but not edit the shape of solid and mesh models; it can apply operations to them, such as copying, moving, rotating, and arraying.



The **EDGESURF** command (short for “edge surface”) constructs a 3D mesh between four *edges*.

The edges of the mesh are defined by open objects: lines, arcs, splines, and open 2D or 3D polylines. The figure below illustrates a mesh that was created between two lines and two arcs. (Technically, AutoCAD interpolates a bicubic mesh between four general space curves, creating a Coons patch.)



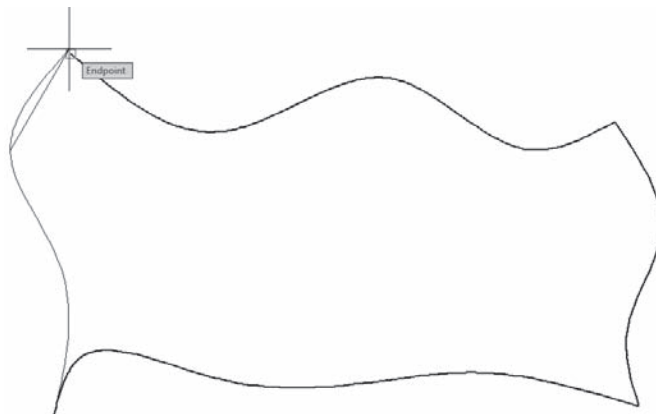
Note For the **EDGESURF** command to work correctly, there must be precisely four objects — no more, no less. Even more important: the four objects *must* connect at their endpoints, so use object snap or snap modes to ensure they do.

(*History:* The “edge surf” name of this command implies that it creates surface objects. But here the word “surface” means that the object is not solidly filled. In earlier releases of AutoCAD, the surface was made of a polyface mesh; as of AutoCAD 2010, the **MESHTYPE** system variable determines whether the surface is made from polyfaces or from 3D mesh objects. This type of surface was developed by Steven Coons, co-developer of the first CAD system. In 1963, he envisioned “the designer seated at the console [early term for keyboard], drawing a sketch of his proposed device on the screen of an oscilloscope tube [early form of monitor], with a light pen modifying his sketch into a perfect drawing.” accad.osu.edu/~waynec/history/lesson4.html.)

TUTORIAL: MESHING BETWEEN FOUR EDGES

In this tutorial, you draw four connected edges.

1. Start AutoCAD with a new drawing (*acad3d.dwt*), and then ensure that **ENDpoint** object snap is turned on.
2. Using the **SPLINE** command, draw four wavy splines, ensuring that their endpoints meet.



Command: **spline**

Specify first point or [Object]: *(Pick a point.)*

Specify next point: *(Pick a point.)*

Specify next point or [Close/Fit tolerance] <start tangent>: *(Pick a point.)*

Specify next point or [Close/Fit tolerance] <start tangent>: *(Pick a point.)*

Specify next point or [Close/Fit tolerance] <start tangent>: *(Press Enter.)*

Specify start tangent: *(Press Enter.)*

Specify end tangent: *(Press Enter.)*

3. To construct surfaces between four objects, start the **EDGESURF** command:
 - In the 3D ribbon's Mesh Modeling tab, choose **EdgeSurf** in the Primitives panel.
 - At the Command: prompt, enter the **edgesurf** command.

Command: **edgesurf** *(Press ENTER.)*

In the command bar, AutoCAD reports the current setting for the **SURFAB1** and **SURFAB2** system variables, which are discussed next.

Current wire frame density: SURFTAB1=6 SURFTAB2=6

4. You are prompted to select the first object. Choose one of the splines.

Select object 1 for surface edge: *(Pick an object.)*

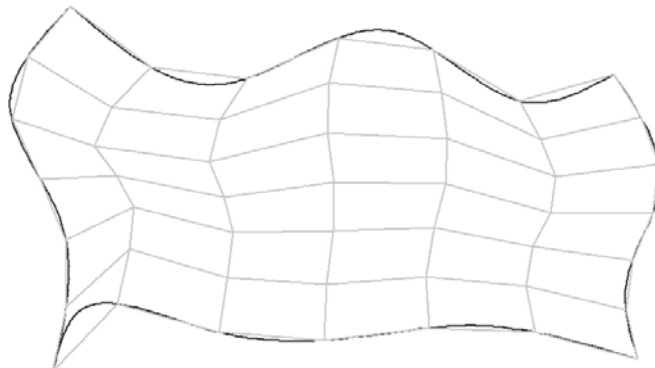
Your selection must consist of a single pick; you cannot take a shortcut, such as windowing all four splines, or pressing **CTRL+A**. The order in which you select the splines, however, does not matter.
5. And then choose the other three splines:

Select object 2 for surface edge: *(Pick object.)*

Select object 3 for surface edge: *(Pick object.)*

Select object 4 for surface edge: *(Pick object.)*

AutoCAD instantly draws the mesh. In the figure below, I've change its color to gray so that you can distinguish it from the four splines that define its borders.



Note If one object is not connected to the adjacent one, AutoCAD complains "Edge x does not touch another edge." You can force objects to touch with the **FILLET** command, or the **TRIM** and **EXTEND** commands; then restart the **EDGESURF** command.

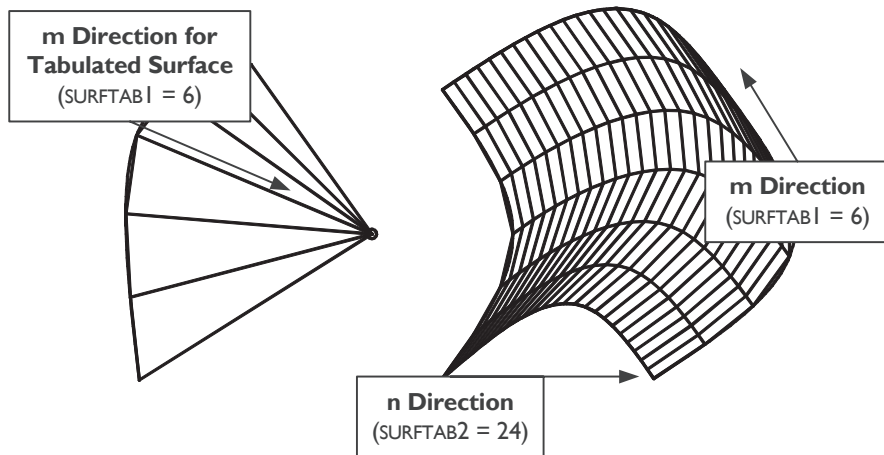
DRAWING SURFACES: ADDITIONAL METHODS

The mesh resulting from the tutorial above is quite coarse, because it uses just 36 faces on the surface area — 6 x 6. You can improve the smoothness with these two system variables.

- **SURFTAB1** sets tabulations and mesh density in the *m* direction.
- **SURFTAB2** sets mesh density in the *n* direction.

What are the *m* and *n* directions? The first object you selected defines the *m* direction; the object connected to the first boundary line defines the *n* direction. The letters “*m*” and “*n*” are used instead of *x* and *y*, because the direction of the meshes doesn’t necessarily correspond with the *x* and *y* axes, and because the two directions are not necessarily at right angles.

These two system variables also affect the smoothness of the surface meshes created by the **REVSURF**, **RULESURF**, and **TABSURF** commands, as illustrated below and discussed later.



As you saw in the tutorial, the default value of 6 for **SURFTAB1** and **SURFTAB2** is way too low. In general, the larger the mesh, the larger the values of these two system variables. I recommend you set the value to at least 16; the range is between 2 and 32766. (The technical editor recommends using 32766 when you need a long lunch break.)

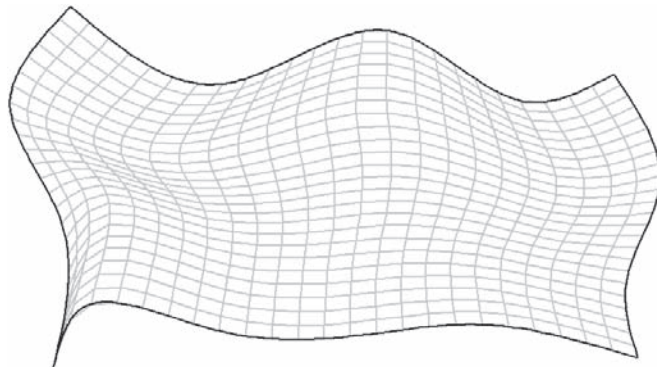
Command: **surftab1**

Enter new value for SURFTAB1 <6>: *(Enter a higher value, such as 24.)*

Command: **surftab2**

Enter new value for SURFTAB2 <6>: *(Enter a higher value, such as 24.)*

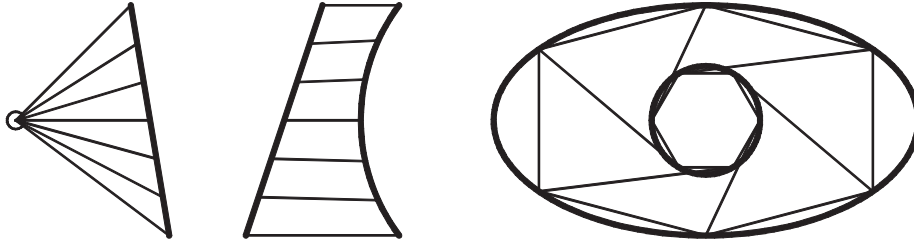
Unfortunately, when you change the value of these system variables, the change is not retroactive. You have to erase the mesh, and then reconstruct it; shown below is the mesh using *m* and *n* values of 24.





RULESURF

The RULESURF command (short for “ruled surfaces”) creates meshes that span two objects — lines, points, arcs, circles, any objects drawn as 2D polylines, and 3D polylines. These objects are called “defining curves.” (*Curve* is a general term in mathematics that includes straight lines or segments, and curves.)



Ruled surfaces:

Left: Between a point and line. **Middle:** Between a line and an arc. **Right:** Between a circle and an ellipse.

Using this command is just like using EDGESURF, except that you work with two curves, and they needn’t touch.

Command: **rulesurf** (Press ENTER.)

Current wire frame density: SURFTAB1=6

Select first defining curve: (Pick object.)

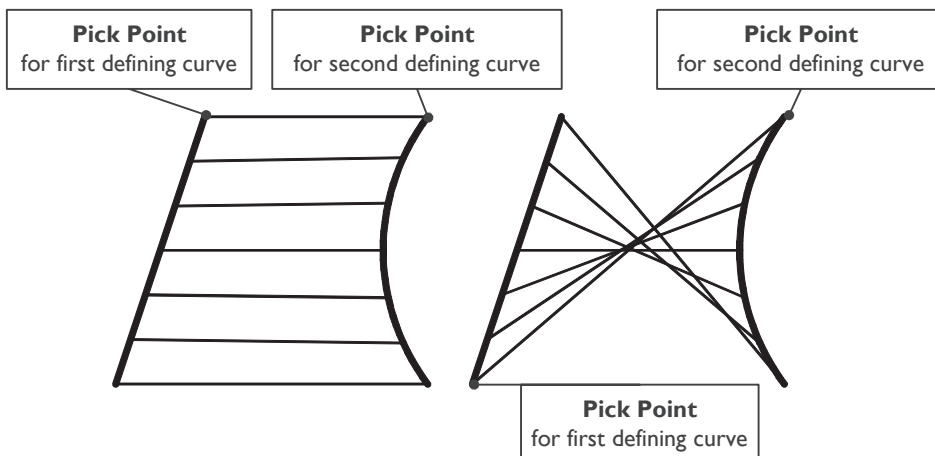
Select second defining curve: (Pick object.)

Defining Starting Points

AutoCAD uses these rules to determine the starting point of the ruled surface:

For Closed Objects. Circles begin at 0 degrees. This angle is affected by the direction of the x axis and the value of SNAPANG. Closed polylines begin at the last vertex, and then go backward to the first vertex. In effect, ruled surfaces are drawn on circles and closed polylines in opposite directions, creating the iris effect illustrated on the previous page. It’s better to use round polylines than circles.

For Open Objects. AutoCAD draws the ruled surface from the endpoint nearest to the pick point on the object. Selecting opposite ends creates ruled surfaces that cross.

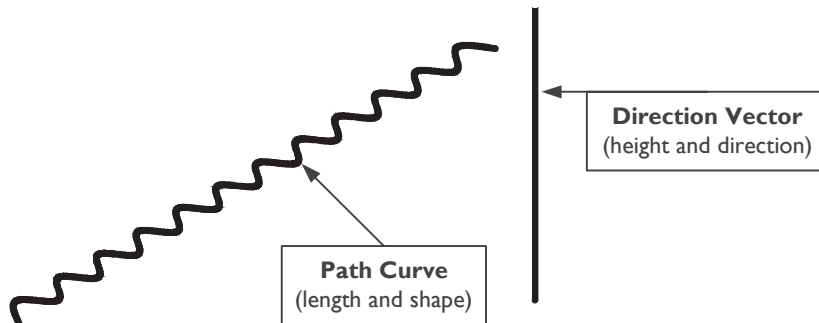




TABSURF

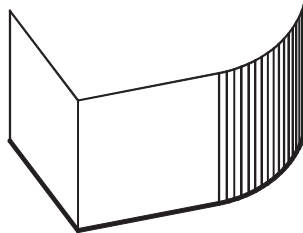
The **TABSURF** command constructs meshes defined by paths and direction vectors. Constructing a tabulated surface takes these steps:

1. Draw a *path curve* from which the mesh is calculated. Path curves can be lines, arcs, circles, ellipses, splines, and 2D or 3D polylines — open or closed.
2. Draw a *direction vector* that defines the direction and length of the mesh. Direction vectors are lines and open 2D or 3D polylines. On polylines, AutoCAD uses only the first and last points to determine the direction vector; intermediate vertices, curves, and splines are ignored. (The direction vector need not touch the path curve, nor must it be perpendicular to the plane of the path curve.)
3. Start the **TABSURF** command, and then pick the path curve and direction vector.



TABSURF needs objects defining its path, direction, and size.

The number of tabulations on the surface is controlled by the **SURFTAB1** system variable. AutoCAD ignores this system variable along polyline segments: tabulation lines are drawn at the ends of straight segments. (Polyline arc segments are divided by the number of tabulations specified by **SURFTAB1**.)



Command: **tabsurf** (Press **ENTER**.)

Current wire frame density: SURFTAB1=6

Select object for path curve: (Pick object.)

Select object for direction vector: (Pick either end of object.)

The point you pick on the direction vector determines the direction that the mesh is projected. Select a point closer to one end of the direction vector, and the tabulation is projected in that direction.



Note To draw vertical lines in plan view, use point filters, as follows:

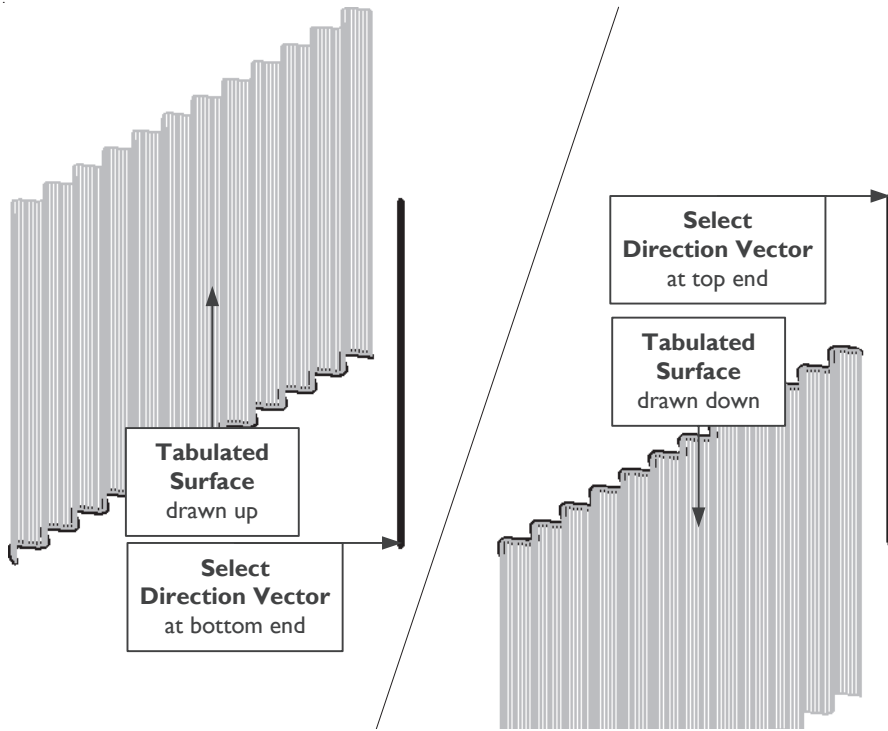
Command: **line**

Specify first point: (Pick a point.)

Specify next point or [Undo]: **.xy**

of (Pick another point, or even the same point.)

(need Z): (Enter a height, such as 10.)

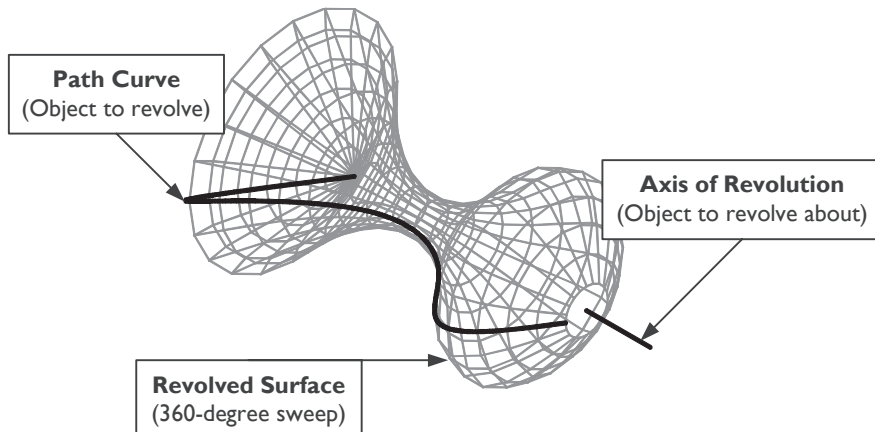


Pick point on the direction vector determining direction of tabulated surface.

REVSURF

The **REVSURF** command (short for “revolved surface”) constructs 3D meshes by rotating objects about an axis. To create the revolved surfaces, you take these steps:

1. Draw a *path curve*, which is the object to be revolved. The path curves can be lines, circles, arcs, splines, and 2D or 3D polylines — they must be open. A sphere, for example, can be made from an arc rotated 360 degrees, while a torus (donut) is a circle that has been rotated through 360 degrees.
2. Draw an *axis of revolution*, about which the object revolves.
3. Use the **REVSURF** command to construct the meshed surface. The axes can be lines and open polylines (2D or 3D); the same considerations apply as with the **TABSURF** command.



This command differs from the previous three in that you specify angles. The start and included angles allow the construction of a revolution of less than 360 degrees (full circle); this lets you see inside the 3D object, like the one illustrated here.

Command: **revsurf** (Press **ENTER**.)

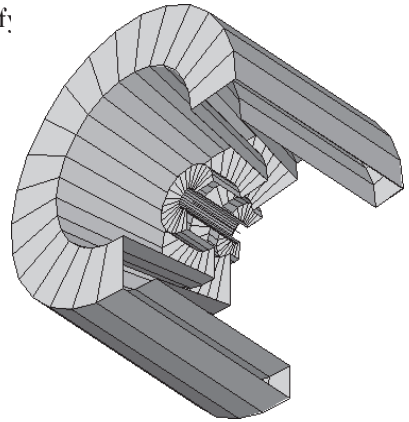
Current wire frame density: SURFTAB1=6 SURFTAB2=6

Select object to revolve: (Pick object.)

Select object that defines the axis of revolution: (Pick object.)

Specify start angle <0>: (Press **ENTER**, or specify an angle, such as **15**.)

Specify included angle (+ccw, -cw) <360>: (Press **ENTER**, or specify an angle, such as **285**.)



The start angle is defined by the location of the path curve, not by the x axis. The right-hand rule determines the direction; specify a negative angle for the surface to revolve in the opposite direction.

(The related **REVOLVE** command creates solid models, as discussed in Chapter 16, “Introduction to 3D Solid Modeling.”)



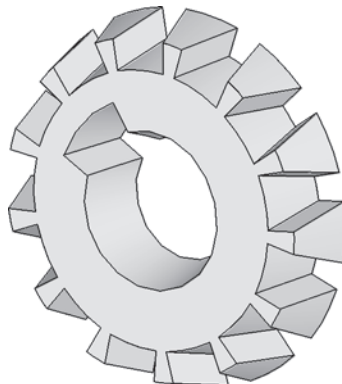
SWEEP AND EXTRUDE

The **SWEEP** and **EXTRUDE** commands extrude 2D objects along paths to create 3D solid models. Both are more sophisticated than both the **PRESSPULL** command from Chapter 16 and the dragging of subobjects from Chapter 17, because you specify a path along which to direct the extrusion. This is handy for creating objects like handrails and snaking tubes.

The two commands operate similarly; **EXTRUDE** has these differences from **SWEEP** :

The extrusion process takes place at right angles to the center line of the path: the profile must be perpendicular to the path; in contrast, the sweep process does not have this restriction. (The solution is to draw the path in one UCS and the profile in a second UCS at an angle to the first. AutoCAD reorients the profile to be perpendicular to the path.)

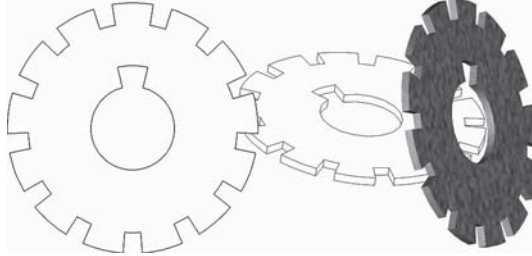
EXTRUDE can specify an angle, so that the extrusion grows smaller or larger along the path, as illustrated below; **SWEEP** cannot do this. (It would be nice if Autodesk would combine the best of both commands into one.)



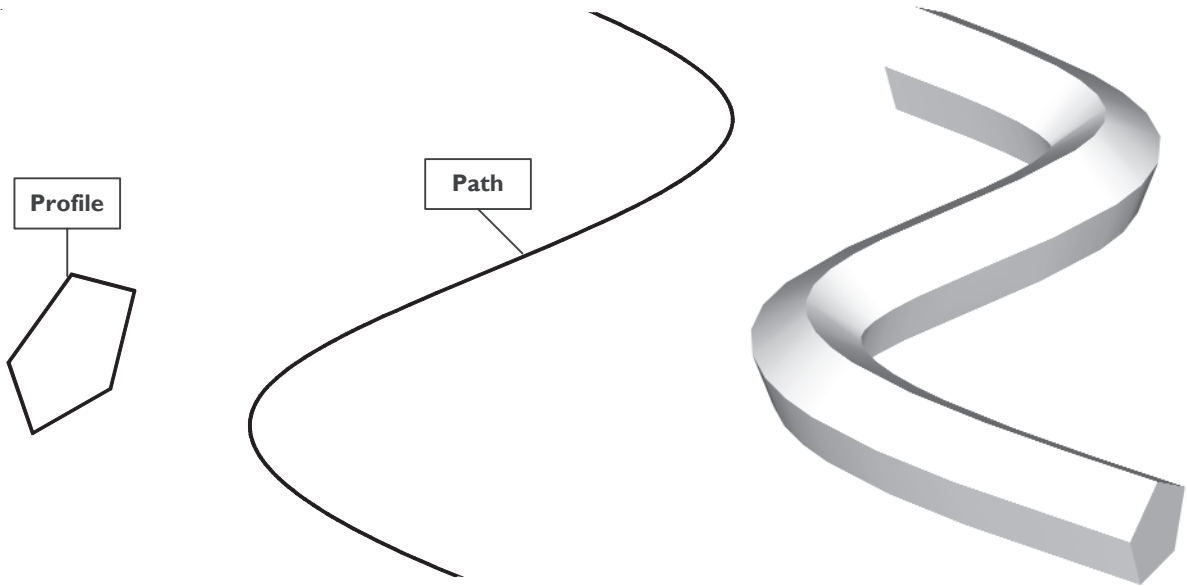
To create extruded or swept models, take these steps:

1. Draw the profile of objects to be extruded or swept. This includes circles, ellipses, closed polylines (including polygons and donuts), closed splines, regions, and 3D faces. AutoCAD cannot extrude/sweep objects within blocks, polylines that cross over themselves, and polylines with more than 500 vertices.

If you plan to extrude objects with holes in them, such as gears with axle holes, then you must convert the objects to a single region. Use the **SUBTRACT** command to remove holes.



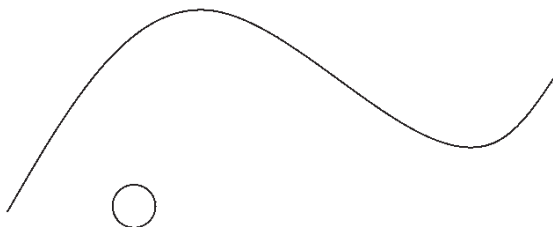
2. Draw the path along which to extrude/sweep. The path is either an object — line, circle, arc, ellipse, elliptical arc, polyline, or spline — or a height that you specify.
3. Use the **EXTRUDE** or **SWEEP** command to create the extrusion.



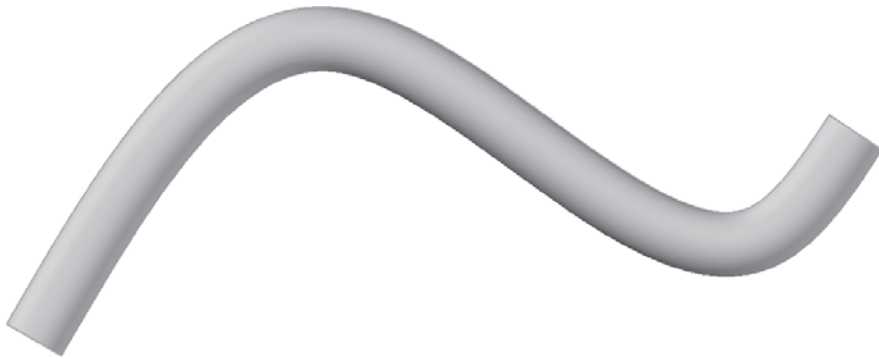
TUTORIAL: SWEEPING OBJECTS INTO SOLIDS

In this tutorial, you sweep a circle along a spline to create a bent handrail shape.

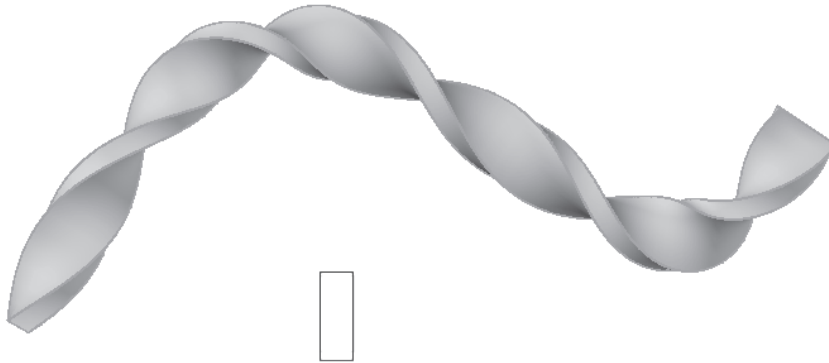
1. Draw a circle; this object defines the profile of the handrail.
2. Draw a spline; this object defines the length (path) of the handrail. The spline and circle do not need to be near each other.



3. To sweep the 2D object into a 3D solid, start the **SWEEP** command:
 - In the 3D ribbon's Home tab, choose **Sweep** from the Modeling panel.
 - At the 'Command:' prompt, enter the **sweep** command.
Command: **sweep** (Press ENTER.)
4. AutoCAD prompts you select the object(s) to sweep.
Current wire frame density: ISOLINES=4
Select objects to sweep: (Pick the circle.)
Select objects to sweep: (Press ENTER to end object selection.)
5. Indicate the height to which the object(s) should be extruded:
Select sweep path or [Alignment/Base point/Scale/Twist]: (Pick the spline.)



As an interesting aside, this command can twist the swept objects with the **Twist** option. AutoCAD asks for the twist angle and whether you want *banking* (in which the swept object is naturally banked along 3D paths). Below, I swept a rectangle along the spline through 1000 degrees.



Note You can sweep subobjects, such as faces and edges of 3D solids and meshes: hold down the **CTRL** key, and then select the subobjects. This command also sweeps multiple profiles along a single path at once, useful for drawing a curving pair of railway tracks

The **EXTRUDE** command operates similarly, except that you don't select a path, but you do get the **Angle of Taper** option:

Command: **extrude** (Press ENTER.)
Current wire frame density: ISOLINES=12
Select objects: (Select one or more objects.)

Select objects: *(Press ENTER to end object selection.)*

Specify height of extrusion or [Path]: *(Enter a value.)*

Specify height of extrusion — pick two points to indicate the height, or enter a distance.

Path — pick a line or other open object. Remember that the path object must be perpendicular to the object being extruded.

Specify angle of taper for extrusion <0>: *(Press ENTER, or enter an angle such as 5.)*

The angle you specify must not allow the extruded solid to intersect itself; thus shallow angles work best.



The **UNION** command merges 2D regions and 3D models into a single body, as you saw earlier in Chapter 16. It also works with 3D mesh objects, including those made with the **MESH** command and with commands like **EDGESURF**.

With mesh objects, however, the process differs somewhat, because AutoCAD turns the meshes into 3D solids. After the union is complete, you turn the 3D solids back into meshes. You see how this works in the following tutorial.



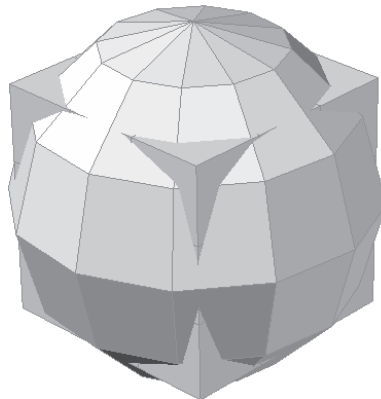
Notes The **UNION**, **INTERSECT**, and **SUBTRACT** commands are collectively known as “Boolean operations,” named after George Boole, a 19th-century mathematician who developed Boolean logic.

You may be familiar with using the terms **or** (union), **and** (intersect), and **not** (subtract) in narrowing down searches on the Internet. These are based on the work of Mr. Boole. You can learn more about this topic at www.kerryr.net/pioneers/boole.htm.

TUTORIAL: JOINING 3D MESHES

In this tutorial, you draw a box and a sphere as 3D mesh objects, and then join them into a single object.

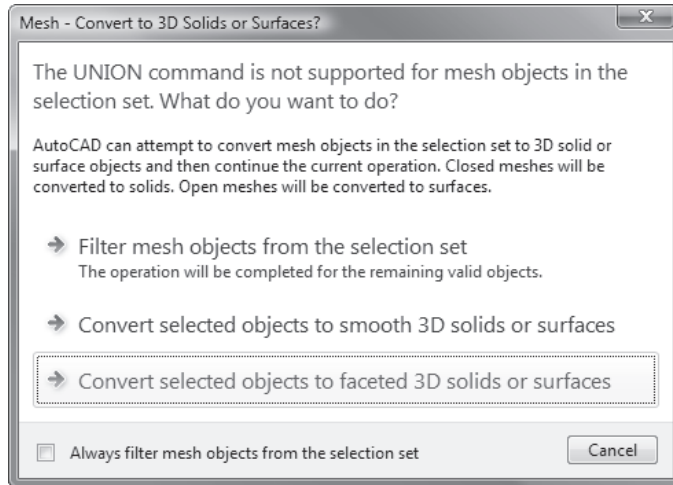
1. In a new drawing, draw a box and a sphere with the **MESH** command, as shown below.



2. To join two or more 3D mesh objects, start the **UNION** command:
 - From the ribbon's Home tab, choose **Union** from the Solid Editing panel.
 - At the 'Command:' prompt, enter the **union** command.

Command: **union** (Press ENTER.)

- Alternatively, enter the alias **uni** at the keyboard.
3. In all cases, AutoCAD prompts you to select the objects to join. Select the two:
 Select objects: (Select the box.)
 Select objects: (Select the sphere.)
 4. Press **Enter** to end object selection:
 Select objects: (Press ENTER to end object selection.)
 5. Notice the dialog box:



AutoCAD gives you three options, the first two of which you don't want:

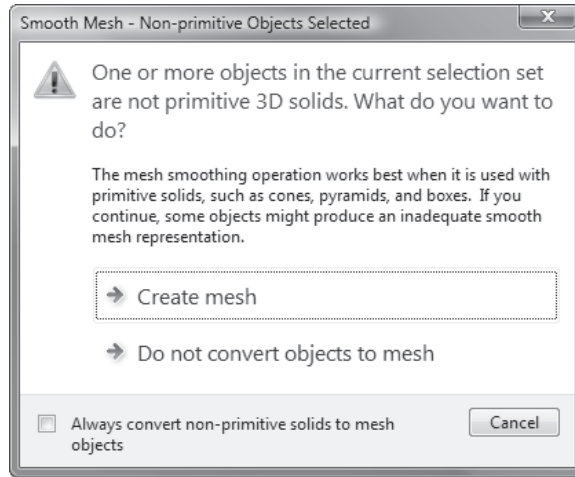
- **Filter mesh objects** — does not convert the mesh objects to solids, and so won't union them.
- **Smooth 3D solids** — does convert the meshes to solids, but into a format that does not convert back well to meshes.
- **Faceted 3D solids** — converts the meshes into a format that converts back.

Choose the third option, **Convert Selected Objects to Faceted 3D Solids or Surfaces**.

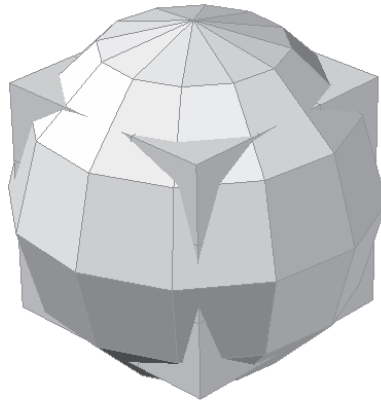
The selected objects are merged into a single object. It does not look any different from before. This is normal.

6. Use the **MESHSMOOTH** command to convert the 3D solid back to a 3D mesh.
 Command: **meshsmooth**
 Select objects to convert: (Select the 3D solid.)
 Select objects to convert: (Press ENTER to end object selection.)

7. After you press **Enter**, AutoCAD displays this dialog box that gives a false warning message — well, false in our case. Recall that we asked AutoCAD to convert the 3D meshes into *faceted* solids; if we'd picked *smoothed* solids, the results would now look atrocious (as illustrated later).
 Choose **Create Mesh**.

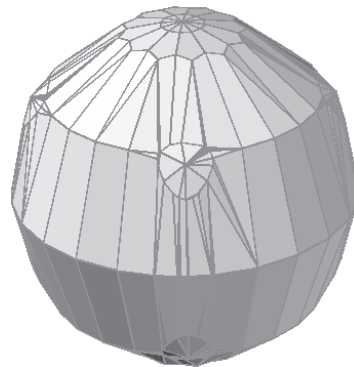


Again, the converted mesh model looks no different, but the **PROPERTIES** command will report now that it is a single mesh object.



Smoothed Conversion

So, how bad does it look when 3D meshes are converted to smooth solids? First, here is the result after conversion to smooth solids (at left):



Left: 3D mesh converted to smooth 3D solid.
Right: 3D solid converted to 3D mesh.

In some cases, this may be exactly what you want. After all, the **U** command lets us play around with what-ifs in AutoCAD. The problem gets worse after you use **MESHSMOOTH** to convert the 3D solid back to mesh form. Have a look (at right, above).

SUBTRACT

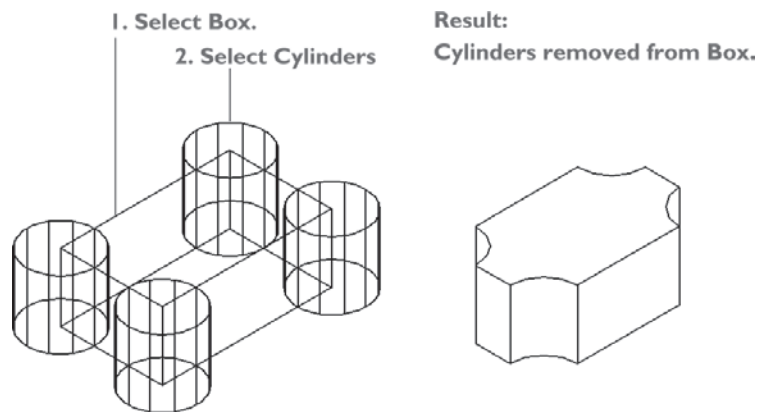
The **SUBTRACT** command removes intersecting parts of 2D regions, 3D solid models, surfaces, and meshes. As with the **UNION** command, AutoCAD must first convert 3D meshes into solids.

Regions, 3D solids, and so on cannot be mixed: like subtracts from like. The objects must be intersecting to be subtracted. AutoCAD needs to know just two things from you: (1) the objects to be subtracted, and (2) the objects doing the subtraction.

TUTORIAL: SUBTRACTING SOLIDS

1. Draw a box (with the **BOX** command) and four cylinders (**CYLINDER** command) similar to those illustrated below.
2. To remove 3D solids from each other, start the **SUBTRACT** command:
 - From the ribbon's Home tab, choose **Subtract** from the Solid Editing panel.
 - At the 'Command:' prompt, enter the **subtract** command.
 - Alternatively, enter the alias **su** at the keyboard.

Command: **subtract** (Press ENTER.)



One result of subtracting the cylinders from the box.

3. In all cases, AutoCAD prompts you to select the objects that will be subtracted.
 Select solids and regions to subtract from ...
 Select objects: (Select one or more objects.)
 Select objects: (Press ENTER to end object selection.)
4. Select the objects that will do the subtracting.
 Select solids and regions to subtract ...
 Select objects: (Select one or more objects.)
 Select objects: (Press ENTER to end object selection.)

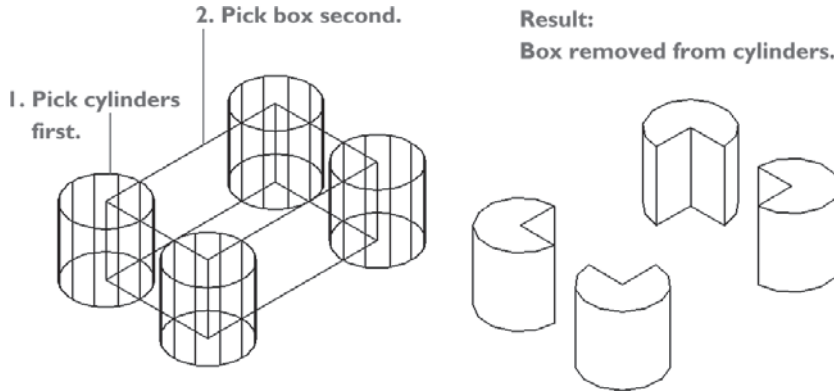


Notes When two or more objects are selected to “subtract from,” they are first joined into a single solid before others are subtracted.

The results may not be what you expect, because the order in which you select objects is important:

1. First, select the objects from which the subtracting takes place. These are the objects to be made smaller.
2. Second, select the objects that do the subtracting.

If the result looks wrong, you selected objects in reverse order. Use the **U** command to undo, and try again.



Another result of subtracting the box from the cylinders.

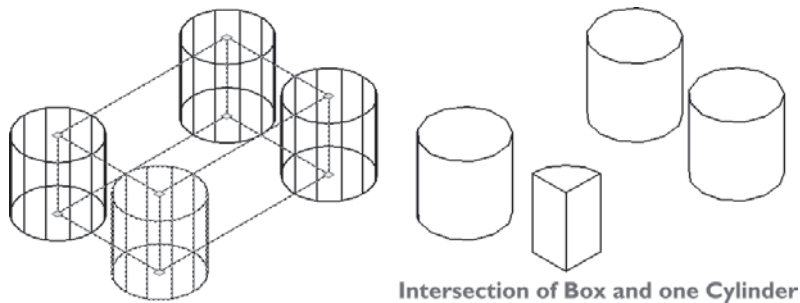
INTERSECT

The **INTERSECT** command removes all but intersecting parts of 2D regions, 3D solids, surfaces, and meshes.

This command determines the overlapping area of regions or the common volume of 3D objects. (Object types, such as solids and regions, cannot be mixed.)

TUTORIAL: INTERSECTING SOLIDS

1. Draw the same cylinders and box, as before.



Intersection of the box and the left, front cylinder resulting in a pie-shaped solid.

2. To remove all but the intersections of the objects, start the **INTERSECT** command:
 - From the ribbon's Home tab, choose **Intersect** from the Solid Editing panel.
 - At the 'Command:' prompt, enter the **intersect** command.

Command: **intersect** (Press **ENTER**.)

- Alternatively, enter the alias **in** at the keyboard.

3. In all cases, AutoCAD prompts you to select the objects from which to make the intersection. For this command to work correctly, the objects solids you select must intersect.

Select objects: *(Select one or more objects.)*

Select objects: *(Press ENTER to end object selection.)*

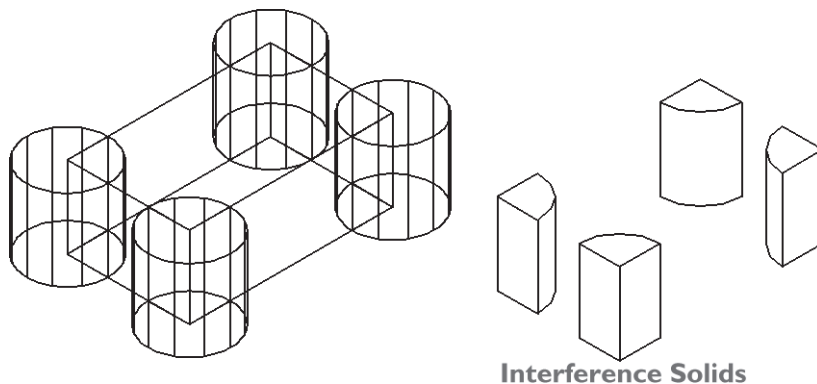
Sometimes the result of this command is — nothing! AutoCAD reports, “Null solid created – deleted.” This occurs when you select objects that do not intersect, such as the box and all four cylinders in the figure above. The four cylinders intersect the box, but not each other. Because this command finds the volume common to all five objects, the result is nothing.



INTERFERE

The INTERFERE command creates 3D bodies from intersecting solids.

This command is more general than the INTERSECT command, because it finds *all* intersections of *all* 3D solids. It deals with interferences in two ways: first, it highlights the solids that interfere (not the interference volumes themselves, unfortunately), and then it gives you the option to turn the interferences into independent solids. You can, if you like, move them away from the rest of the solid model, and then analyze them separately.



Left: Common volumes of solids found with the **INTERFERE** command.

Right: The interfering solids moved from the original solids.

I think you can see that these two features are immediately useful. By highlighting interfering pairs of solids, AutoCAD shows you which parts of a model collide or interfere — for example, pipes that go through other pipes when they ought not to, or heating ducts that penetrate walls.

TUTORIAL: CHECKING INTERFERENCE BETWEEN SOLIDS

1. To determine if 3D solids interfere with each other, start the **INTERFERE** command:
 - From the ribbon's Home tab, choose **Intersect** from the Solid Editing panel.
 - At the 'Command:' prompt, enter the **interfere** command.
Command: **interfere** *(Press ENTER.)*
 - Alternatively, enter the alias **inf** at the keyboard.

2. AutoCAD prompts you to select two sets of solids:

Select first set of solids: *(Select one or more objects.)*

Select objects: *(Press ENTER to end object selection.)*

The prompts provide no clue that there is an alternative method to select objects. At the first prompt, enter **all** to select all objects in the drawing, and then press **ENTER** at the second prompt. AutoCAD filters out all non-solids, and then checks them against each other for interference.

3. Optionally, select a second set of objects.

Select second set of solids: *(Press ENTER to skip, or select objects for the second selection set.)*

AutoCAD compares the solids against each other, looking for interferences, producing a report that looks similar to the following:

Comparing 1 solid against 4 solids.

Interfering solids (first set): 1

(second set): 4

Interfering pairs : 4

4. Optionally, AutoCAD can create solids out of the volumes that interfere (intersect).

Create interference solids? [Yes/No] <N>: **y**

5. If you want, AutoCAD can highlight pairs of interfering objects. This is useful when three or more objects interfere.

Highlight pairs of interfering solids? [Yes/No] <N>: **y**

6. Press **N** to see the next pair of interfering solids.

Enter an option [Next pair/eXit] <Next>: **n**

7. When finished checking, enter **x** to exit the command:

Enter an option [Next pair/eXit] <Next>: **x**

At this point, the model may look no different. To isolate the newly-created solids representing the interference volumes, use the **MOVE** command together with the **Previous** option. (When you selected the solids originally at the “Select first/second set of solids” prompt, AutoCAD placed them into the Previous selection set.)

Command: **move**

Select objects: **p**

Select objects: *(Press ENTER.)*

Specify base point or displacement: *(Pick a point.)*

Specify second point of displacement or <use first point as displacement>: *(Pick another point.)*

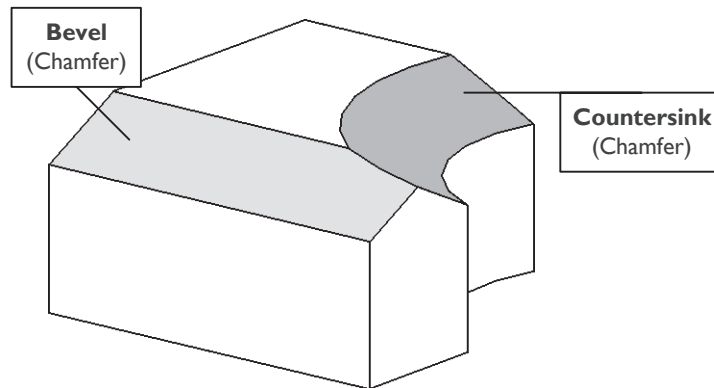
This moves the original solids away from the interference solids. The result is similar to the figure on the previous page.

CHAMFER

The **CHAMFER** command chamfers the edges of 3D objects, in addition to the intersections of 2D objects.

When you select a 3D solid or mesh model, this command automatically switches to 3D editing mode that presents prompts different from those you saw in 2D editing. This command operates on a single 3D model at a time, but allows you to chamfer as many edges as you need during a single operation. (It does not work with surface models.)

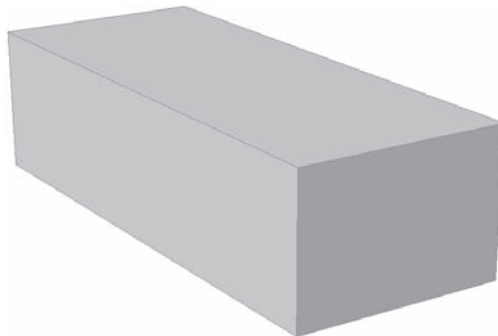
This command also works on curved portions of solid models. You can pick an isoline, the line that defines curved surfaces. When you select the isoline of a hole, for example, then AutoCAD creates a countersink.



TUTORIAL: CHAMFERING OBJECTS

In this tutorial, you draw a rectangular box, and then chamfer its edges.

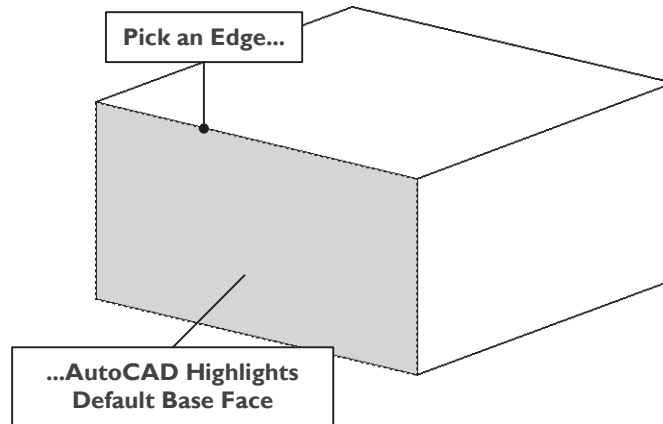
1. Draw a rectangular box with the **BOX** command.



2. To chamfer the edge (intersection of two faces) of the solid model, start the **CHAMFER** command:
 - In the ribbon's Home tab, choose **Chamfer** from the Modify panel.
 - At the 'Command:' prompt, enter the **chamfer** command.
Command: **chamfer** (Press ENTER.)
 - Alternatively, enter the **cha** alias at the 'Command:' prompt.
3. In all cases, AutoCAD displays prompts familiar to you from 2D editing:
(TRIM mode) Current chamfer Dist1 = 0.0000, Dist2 = 0.0000
Select first line or [Polyline/Distance/Angle/Trim/Method/mUltiple]: (Pick a solid model.)

- **Select an Edge** — AutoCAD does two things: (1) highlights one face by outlining its edges with dashed lines, and (2) prompts you to select the *base surface* (a.k.a. the first face). The first chamfer distance applies to the base face, which AutoCAD calls the “base surface chamfer distance.”

Because the two chamfer distances can be unequal, AutoCAD needs to know which distance applies to which face. (Recall that every edge has two adjoining faces.)



- **Or, select an isoline**, and AutoCAD reverses the order: first it asks for the chamfer distances, and then for the other face.

Notice that AutoCAD highlights one of the two surfaces adjoining the selected edge. The following prompt allows you to select a base face other than the one highlighted:

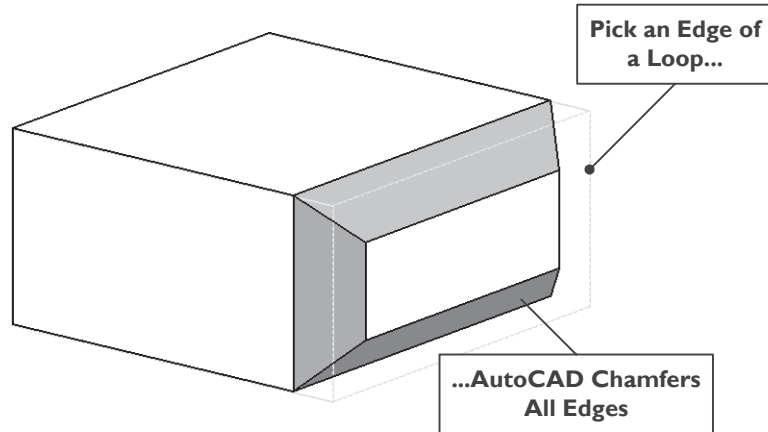
Base surface selection...

Enter surface selection option [Next/OK (current)] <OK>: (Type **N** or **OK**.)

- Enter **N** (next) to select the other face as the base surface.
 - Enter **OK** when the two chamfer distances are the same, or when AutoCAD correctly guesses the base surface.
4. Provide the chamfer distance(s); you cannot specify an angle, as with 2D chamfering. Notice the reference to the “base surface chamfer distance”:
Specify base surface chamfer distance: **0.5**
Specify other surface chamfer distance <0.5000>: (Press **ENTER**, or enter a value.)

If the chamfer distance is too large, AutoCAD complains, “Cannot blend edge with unselected adjacent tangent edge. Finding connected blend set failed. Failure while chamfering,” and exits the command. Restart the command, and enter a smaller value.

5. AutoCAD can chamfer just the selected edge, or a *loop*. A “loop” consists of all the edges that touch the base surface. A rectangular surface, as illustrated below, has a loop with four edges.
Select an edge or [Loop]: (Pick an edge, or type **L**.)
Select an edge or [Loop]: (Press **esc**.)



6. The command repeats until you press **ESC**.

Chamfering Meshes

When you use the **CHAMFER** command on 3D mesh objects, AutoCAD will change them to 3D solids. Remember to follow the same process as described under the **UNION** command:

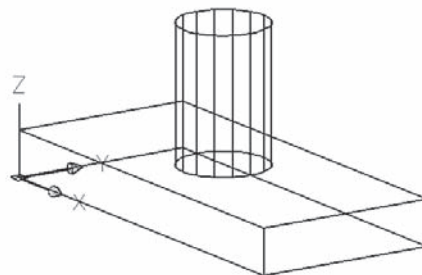
1. Upon AutoCAD's prompting, change the 3D meshes to faceted solids (no smoothed solids).
2. Use **MESHSMOOTH** to convert back to 3D mesh objects.

Fillet

Applying the **FILLET** command to 3D objects is similar to using the **CHAMFER** command, except that it creates rounded corners. AutoCAD is limited to constant radius fillets; it cannot apply a variable radius to fillets.

TUTORIAL: CHAMFERING AND FILLETING 3D OBJECTS

1. In this tutorial, you draw a cylinder on top of a rectangular box, and then add fillets. Begin by drawing the box and cylinder, as follows:



Command: **box**

Specify corner of box or [CEnter] <0,0,0>: (Press **ENTER**.)

Specify corner or [Cube/Length]: **l**

Specify length: **9**

Specify width: **4**

Specify height: **1**

Command: **cylinder**

Current wire frame density: ISOLINES=16

Specify center point for base of cylinder or [Elliptical] <0,0,0>: **4.5,2,1**

Specify radius for base of cylinder or [Diameter]: **1**

Specify height of cylinder or [Center of other end]: **3**

2. Start the **FILLET** command, and fillet the four edges of the end of the box:

Command: **fillet**

Current settings: Mode = TRIM, Radius = 0.0

Select first object or [Undo/Polyline/Radius/Trim/Multiple]: *(Pick an edge.)*

Enter fillet radius <0.0>: **0.2**

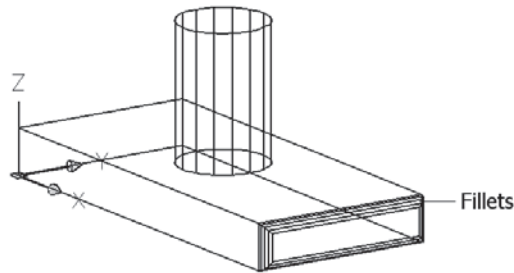
Select an edge or [Chain/Radius]: *(Pick another edge.)*

Select an edge or [Chain/Radius]: *(Pick a third edge.)*

Select an edge or [Chain/Radius]: *(Pick a fourth edge.)*

Select an edge or [Chain/Radius]: *(Press ENTER.)*

4 edge(s) selected for fillet.



3. Another fillet is to be placed at the base of the cylinder. But the **FILLET** command cannot fillet between two solid objects. The procedure is to use the **UNION** command to join the box and the cylinder into a single object.

Command: **union**

Select objects: *(Pick the box.)*

1 found Select objects: *(Pick the cylinder.)*

1 found, 2 total Select objects: *(Press ENTER.)*

The objects won't look any different.

4. Apply the fillet between the box and the cylinder:

Command: **fillet**

Current settings: Mode = TRIM, Radius = 0.2

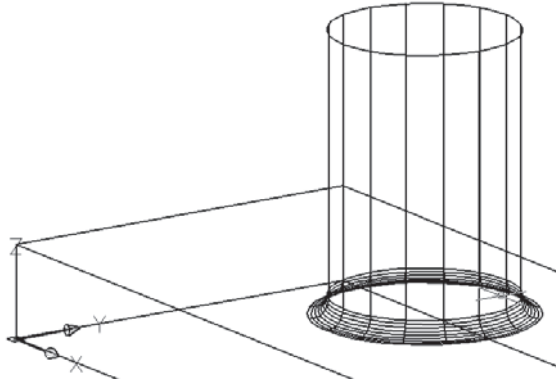
Select first object or [Undo/Polyline/Radius/Trim/Multiple]: *(Pick the bottom edge of the cylinder.)*

Enter fillet radius <0.2>: *(Press ENTER to reuse the same radius.)*

Select an edge or [Chain/Radius]: *(Press ENTER to exit the command.)*

1 edge(s) selected for fillet.

Notice the curve at the base of the cylinder.



5. Apply one last fillet, on top of the cylinder.

Command: **fillet**

Current settings: Mode = TRIM, Radius = 0.2

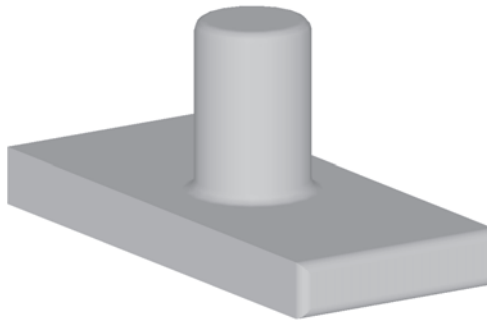
Select first object or [Undo/Polyline/Radius/Trim/Multiple]: *(Pick the top edge of the cylinder.)*

Enter fillet radius <0.2>: *(Press ENTER to reuse the radius.)*

Select an edge or [Chain/Radius]: *(Press ENTER.)*

1 edge(s) selected for fillet.

Notice that the top of the cylinder is gracefully curved.



MASSPROP

The **MASSPROP** command reports information about regions and solid models, such as their area, perimeter, centroid, and so on (short for “mass properties”). This command does not work with 3D meshes or regions.

TUTORIAL: REPORTING PROPERTIES

1. To find information about solids, start the **MASSPROP** command:
 - At the ‘Command:’ prompt, enter the **massprop** command.

Command: **massprop** *(Press ENTER.)*

2. AutoCAD prompts you to select objects. While you can select more than one object, it is better to pick just one, because of the amount of information generated.

Select objects: *(Select solid object.)*

Select objects: *(Press ENTER to end object selection.)*

AutoCAD switches to the Text window, and displays the report:

```

----- SOLIDS -----
Mass:           4114.1195
Volume:         4114.1195
Bounding box:   X: -20.4761 -- 21.5163
                Y: 11.0798 -- 20.7801
                Z: -60.3141 -- -18.3141
Centroid:       X: 1.5929
                Y: 13.5965
                Z: -39.9397
Moments of inertia: X: 7608951.3091
                  Y: 7078292.1662
                  Z: 1020452.9262
Products of inertia: XY: 83775.7436
                  YZ: -2231040.1682
                  ZX: -230014.1591
Radii of gyration: X: 43.0055
                  Y: 41.4788
                  Z: 15.7492
Principal moments and X-Y-Z directions about centroid:
I: 303884.0429 along [0.8643 -0.0306 -0.5021]
J: 505287.2270 along [0.0265 0.9995 -0.0154]
K: 231018.9604 along [0.5023 0.0000 0.8647]

```

The data are relative to the current UCS. Unfortunately, the mass is always equal to the volume because AutoCAD does not allow the density to be anything other than 1.0 — contrary to what we would expect from this kind of command.

3. AutoCAD asks if you wish to save the data to a file.
Write analysis to a file? [Yes/No] <N>: **y**

If you answer **Y**, AutoCAD displays the Create Mass and Area Properties File dialog box. Enter a file name, and then click **Save**. AutoCAD creates the *.mpr* mass properties report file.

4. Press **F2** to return to the drawing window.

MASSPROP DEFINITIONS

Bounding Box is the x, y, z-coordinate triple of the smallest 3D box enclosing 3D solids or bodies.

Centroid is the center of mass of 3D solids and bodies.

Mass equals the volume, because density = 1.

Moment of Inertia is $\text{Mass} * \text{Radius}^2$ for 3D bodies.

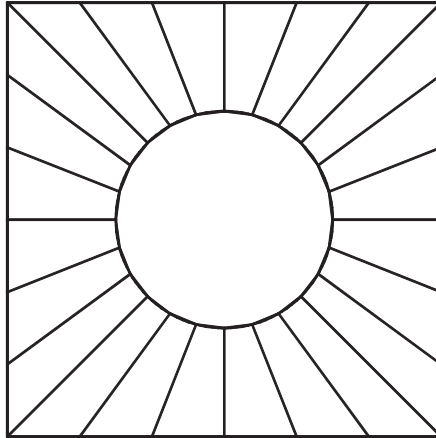
Product of Inertia is $\text{Mass} * \text{Distance (centroid to y, z axis)} * \text{Distance (centroid to x, z axis)}$ for 3D bodies.

Radius of Gyration is $(\text{MomentOfInertia} / \text{Mass})^{1/2}$ for 3D bodies.

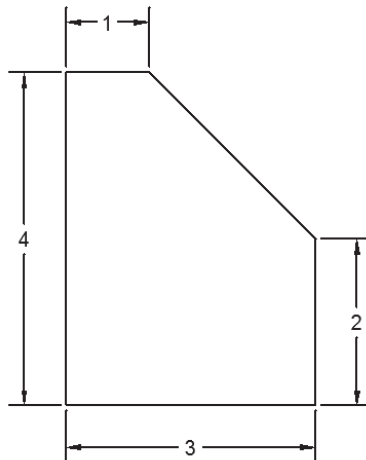
Volume is the 3D space occupied by 3D bodies.

EXERCISES

1. Draw four connected splines using the **SPLINE** command.
Apply a Coons patch between them.
Which command did you use?
2. Use the **REVSURF** command to draw a sphere with a diameter of 2.5 units.
What object do you use as the defining curve?
3. Use the **TABSURF** command to draw a mesh with a hole in it, as illustrated below.
Hint: The defining curves are both polylines.
How must you draw the polylines to keep the tabulations from twisting?



4. Draw the chamfered 12mm-radius washer using the profile illustrated below.
With which command did you draw it?



Use these dimensions:

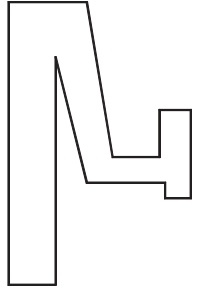
Outer height = 4mm

Inner height = 2mm

Width at bottom = 3mm

Width at top = 1mm

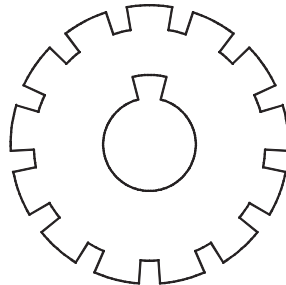
5. From the CD, open the *Ch10RevSurf.dwg* file, a partial drawing of a plastic wheel. Use the **REVSURF** command to generate a 360-degree 3D surface model.



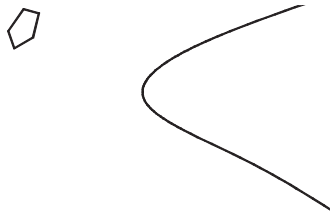
6. From the CD, open the *Ch11Glass.dwg* drawing file, a profile of a goblet. Use the **REVOLVE** command to create the goblet.



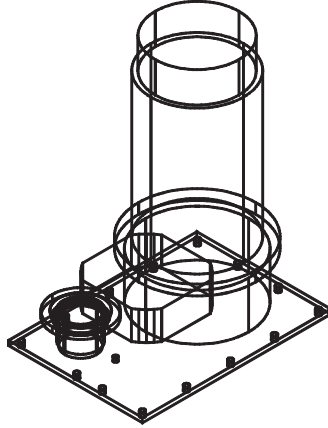
7. From the CD, open the *Ch11Extrude.dwg* drawing file, a profile of a gear.
 - a. Use the **EXTRUDE** command to make the gear 0.5 units thick.
 - b. Repeat, but with a taper angle of 15 degrees.



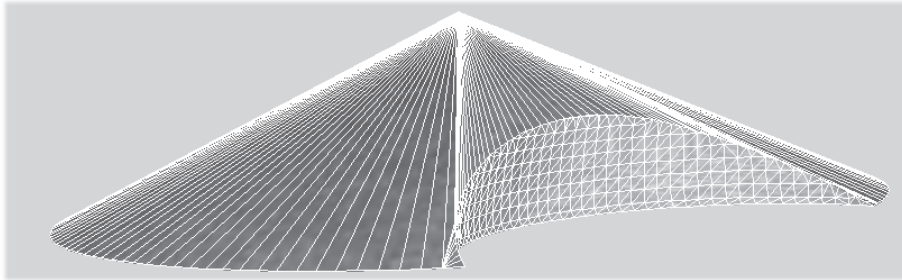
8. From the CD, open the *Ch11Path.dwg* drawing file, a pentagon profile and a path. Use the **SWEEP** command to sweep the pentagon along the spline path.



9. Create a profile of your own, and then apply the **SWEEP** command.
10. From the CD, open the *Ch11Pump.dwg* file, a drawing of a pump.
Use the **INTERFERE** and **MOVE** commands to isolate the interference solids.



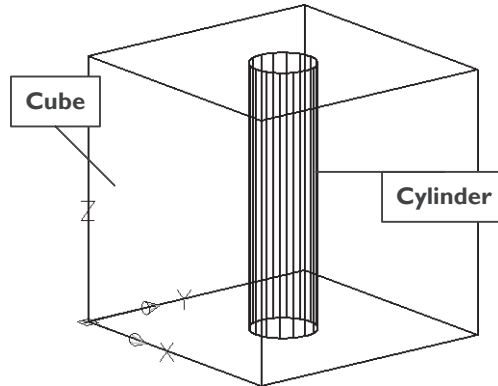
11. From the CD, open the *Ch11Sandpile.dwg* file, a drawing of a pile of sand, from which a bulldozer removed a chunk.



- a. Use the **MASSPROP** command to find the volume of sand (in cubic feet) in the pile.
- b. Assuming sand weighs 90 pounds per cubic foot, how much does the pile weigh?

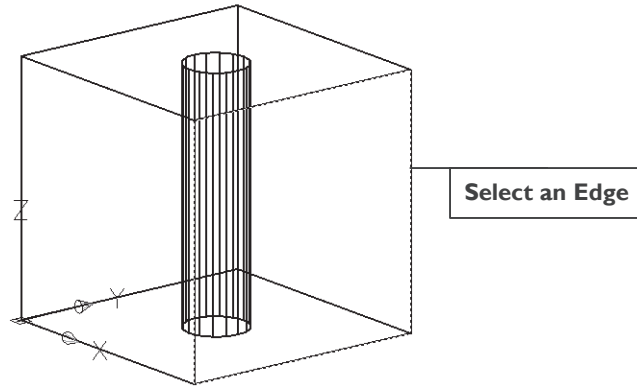
ADDING BEVELS TO SOLIDS' EDGES

Before applying the CHAMFER command, first create a simple solid model of a hole inside a cube.



1. Start AutoCAD with a new drawing.
2. Draw a cube, as follows:
 Command: **box**
 Specify corner of box or [CEnter] <0,0,0>: (Press ENTER.)
 Specify corner or [Cube/Length]: **c**
 Specify length: **1**
3. Add the cylinder, as follows:
 Command: **cylinder**
 Current wire frame density: ISOLINES=16
 Specify center point for base of cylinder or [Elliptical] <0,0,0>: **0.5,0.5,0**
 Specify radius for base of cylinder or [Diameter]: **0.125**
 Specify height of cylinder or [Center of other end]: **1**
4. To view the cube better, rotate the viewpoint, as follows:
 Command: **vpoint**
 Current view direction: VIEWDIR= 0,0,0
 Specify a view point or [Rotate] <display compass and tripod>: **2.5,-2,1**
 Regenerating model.
 (To further improve the image, click the **LWT** button to thicken the lines, set **ISOLINES** to **16**, zoom to the extents, and then **ZOOM 0.5x**.)
5. Subtract the cylinder from the box to create the hole:
 Command: **subtract**
 Select solids and regions to subtract from ..
 Select objects: (Pick the cube.)
 1 found Select objects: (Press ENTER.)
 Select solids and regions to subtract ...
 Select objects: (Pick the cylinder.)
 1 found Select objects: (Press ENTER.)
 (Following the subtraction operation, the model will look exactly the same.)

6. Start the **CHAMFER** command, and then select an edge of the cube.



Command: **chamfer**

(TRIM mode) Current chamfer Dist1 = 0.5000, Dist2 = 0.5000

Select first line or [Polyline/Distance/Angle/Trim/Method]: *(Pick any edge of the cube.)*

Base surface selection...

(To highlight a different face, enter **N** — short for “next” — at the prompt; otherwise press **ENTER**.)

Enter surface selection option [Next/OK (current)] <OK>: *(Press ENTER.)*

7. Specify the chamfer distances here. (The **Distance** option earlier in this command is only for 2D entities.)

Specify base surface chamfer distance <0.5000>: **0.2**

Specify other surface chamfer distance <0.5000>: **0.2**

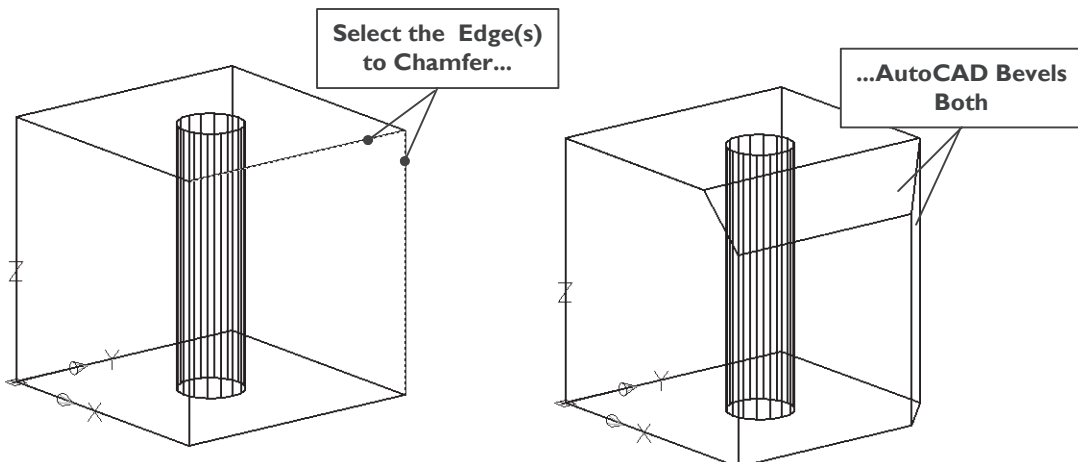
8. Select the edges you want chamfered: pick any two edges, and then press **ENTER** to exit the command.

Select an edge or [Loop]: *(Pick edge 1.)*

Select an edge or [Loop]: *(Pick edge 2.)*

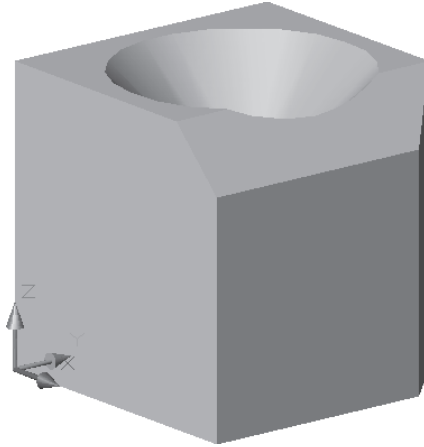
Select an edge or [Loop]: *(Press ENTER.)*

Notice that AutoCAD performs a “double” chamfer: the intersection between the two chamfered edges is another 45-degree angle.



ADDING COUNTERSINKS

In this tutorial, you continue from the previous one to create a second type of chamfer, one that countersinks the hole.



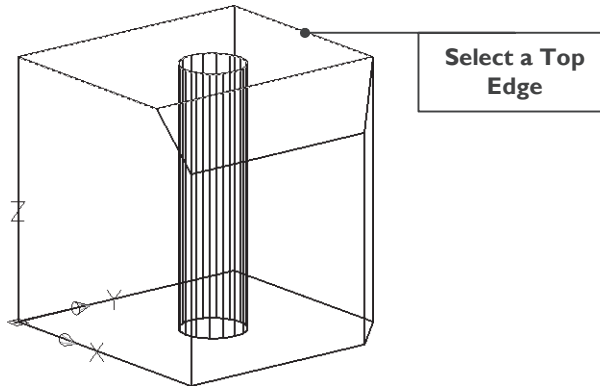
Continue with the drawing from the previous tutorial.

1. Start the **CHAMFER** command again, and then select a top edge on the cube (adjacent to the hole).

Command: **chamfer**

(TRIM mode) Current chamfer Dist1 = 0.2000, Dist2 = 0.2000

Select first line or [Polyline/Distance/Angle/Trim/Method]: *(Pick an edge on the top face of the cube, but not one of the chamfer edges.)*



If the “wrong” face is highlighted by AutoCAD, use the **Next** option to pick the top face:

Enter surface selection option [Next/OK (current)] <OK>: **n**

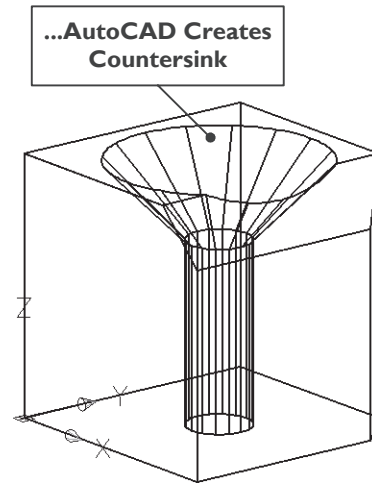
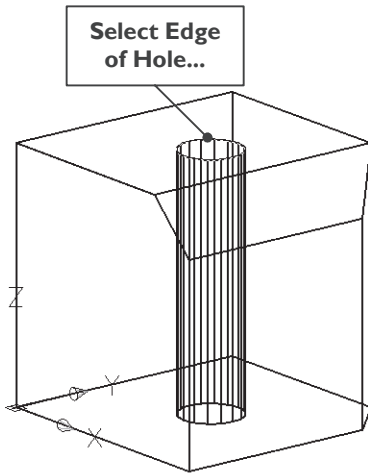
Enter surface selection option [Next/OK (current)] <OK>: *(Press ENTER.)*

2. Specify a chamfer distance of 0.3 units:
 - Specify base surface chamfer distance <0.2000>: **0.3**
 - Specify other surface chamfer distance <0.2000>: **0.3**

3. Select the top of the cylindrical hole:

Select an edge or [Loop]: *(Select the top of the cylindrical hole.)*

Select an edge or [Loop]: *(Press ENTER.)*



CHAPTER REVIEW

1. What is an *extrusion*?
2. Describe two ways that extrusions differ from sweeps:
 - a.
 - b.
3. Describe a use for the following commands in constructing surfaces:
RULESURF
EDGESURF
TABSURF
REVSURF
4. The density of two-direction 3D surfaces is described by the values of m and n . Which system variables control the values of each?
5. Which command creates “Coons patches”?
6. How do **TABSURF** and **REVSURF** differ?
How are they the same?
7. Can chamfer operate on 3D mesh models?
If so, describe the process:
8. Does the **INTERSECT** command remove everything in common from two solid objects?
9. Does the **UNION** command combine two solid objects to create a region?
10. Why are the **UNION**, **INTERSECT**, and **SUBTRACT** commands called “Booleans”?
11. Describe how to create bolt holes in solid objects.
13. Does the **INTERFERE** command create solid objects from the intersection of two or more solid objects?



Plotting Drawings

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CHAPTER 19

Working with Layouts

Up until now, you have been drawing and editing drawings in model space. *Models* are drawn in *model* space; drawings are *laid* out for plotting in *layouts*. Layouts determine the arrangement and views of 2D and 3D models on the plot sheet.

In this chapter, you find the following commands and options:

TILEMODE switches drawings between model and layout tabs.

ZOOM Xp scales models relative to layouts.

LAYER Current VP Freeze freezes layers in selected viewports.

Viewport overrides override layer properties on a per-viewport basis.

VPMAX and **VPMIN** maximize and minimize the active viewport.

QVLAYOUT displays thumbnail views of layouts in filmroll fashion.

SPACETRANS scales model-space distances relative to layouts.

PSLTSCALE scales model-space linetypes relative to layouts.

LAYOUTWIZARD steps through the layout creation process.

LAYOUT creates and modifies layouts.

VIEWPORTS creates rectangular viewports.

-VPORIS and **VPCLIP** create and change polygonal viewports.

EXPORTLAYOUT exports layouts to model space.

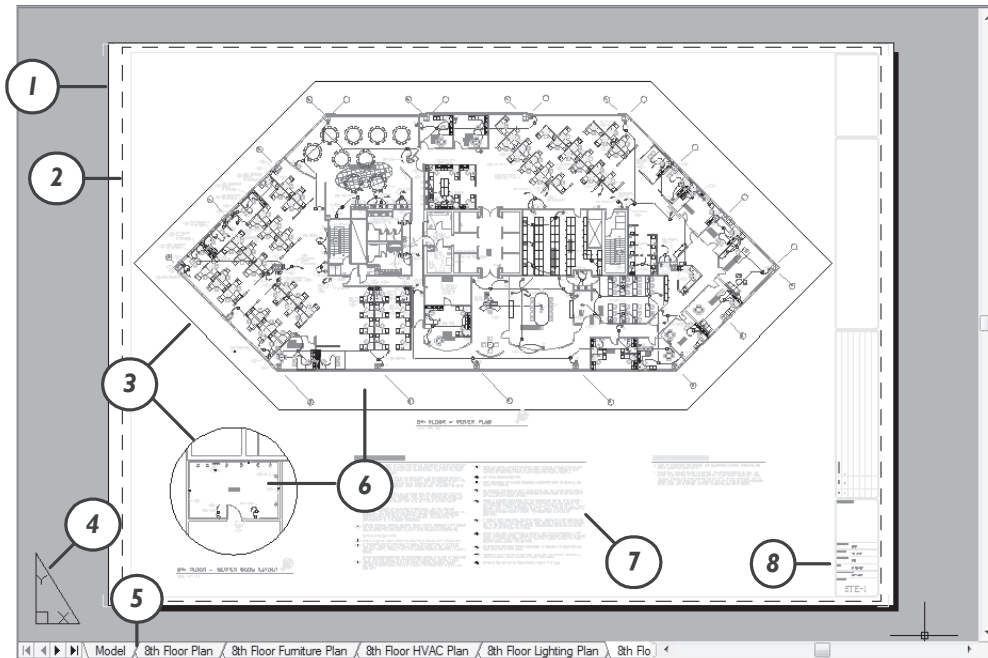
ABOUT LAYOUTS

Two of AutoCAD's drafting environments are known as model space and layouts.

Model space is where you draw the models full scale, 1:1. Recall from earlier chapters that unless you used annotative scaling, you had to calculate manually the correct scale factor for text, linetypes, and hatch patterns. Whether you employed annotative scaling or worked out the scale factors on your own, all this was to ensure text and other elements did not appear too small when printed.

Layouts solve the scaling problem. In this environment, you arrange models as if they were placed on sheets of paper. In layouts, the paper is full size (1:1); instead, the model is scaled down to fit the paper. This solves the problem of working out scale factors for text, linetypes, and hatch patterns — they are all drawn full size in layouts.

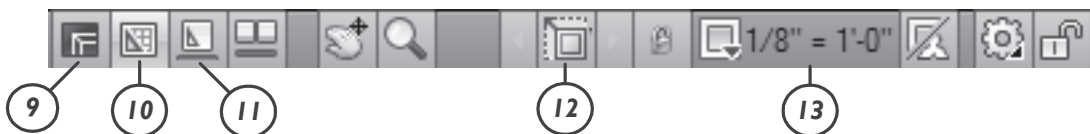
The figure below illustrates a layout with two viewports showing the model at two scales.



- 1: Edge of "paper" (solid line)
- 2: Printer margins (dashed line).
- 3: Polygonal and round viewports.
- 4: Paper space UCS icon.
- 5: Model and layout tabs.
- 6: The model (scaled).
- 7: Text placed in layout.
- 8: Title block placed in layout.

Showing models in layouts takes two steps: (1) creating *viewports* in layouts, which act like windows into model space, and then (2) setting the viewport scale factor to scale the model.

You access layouts through tabs or buttons — depending on how you have AutoCAD configured. Sometimes, layout tabs are located just under the drawing, as illustrated above. Other times, layout viewing controls are stored in buttons on the status bar, as illustrated below.



- 9: Switch to model space.
- 10: Switch to last layout.
- 11: Quick views of layouts.
- 12: Maximize the viewport.
- 13: Viewport scale factor.

Every drawing starts with one model space (named “Model”) and one or more layouts named “Layout1,” “Layout2,” and so on. Each layout represents a drawing sheet to be plotted/printed. Each layout can have the same or different paper sizes and orientation; each can be assigned the same or a different printer.

You can think of layouts as *interactive plot previews*. You can create, move, and delete viewports, the entire model, and other elements at your whim. Layouts show the overall plan as well as zoomed-in details — all at the same time.

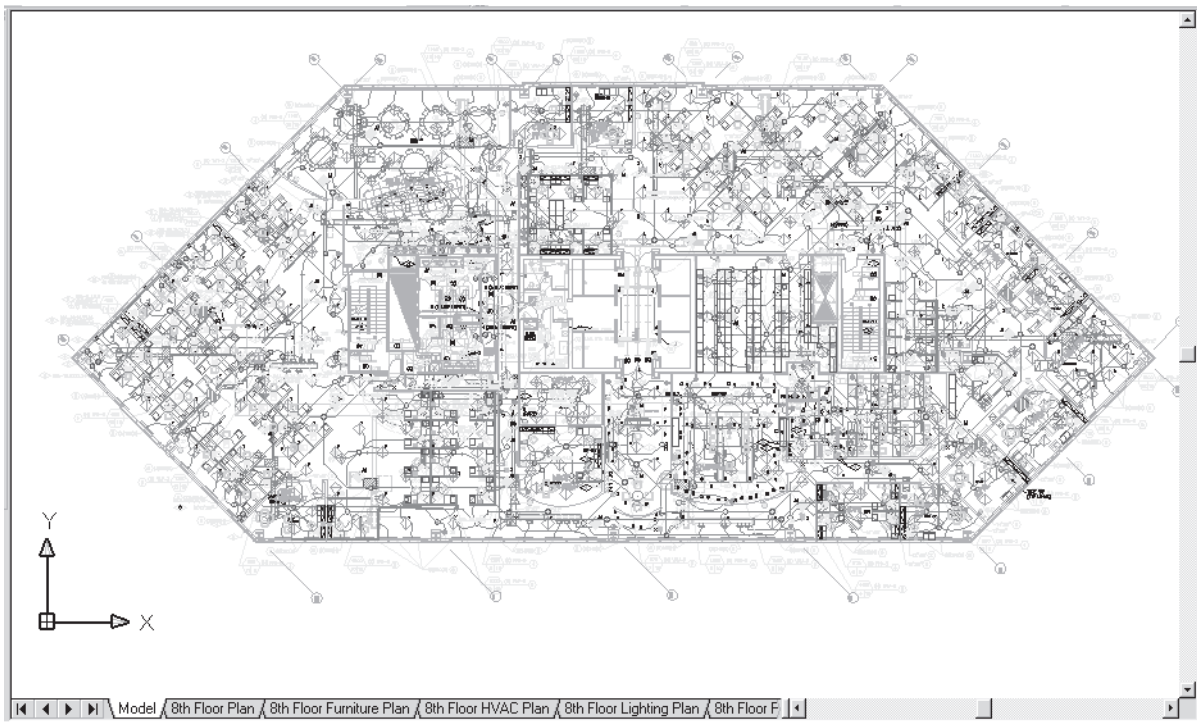
Each layout needs at least one viewport through which shows some or all of the model. But layouts often have two or more viewports, each showing details or overall views. Each viewport can show the model at a different scale factor, and, in 3D drawings, show different views, such as the top, side, and front. (Layouts themselves are strictly 2D.)

Layouts typically have drawing borders and title blocks. (The viewports fit inside the drawing border.) You can add dimensions, text, and hatch patterns to layouts. A command is available that automatically translates linetype scales between the model scale factor and the layout.

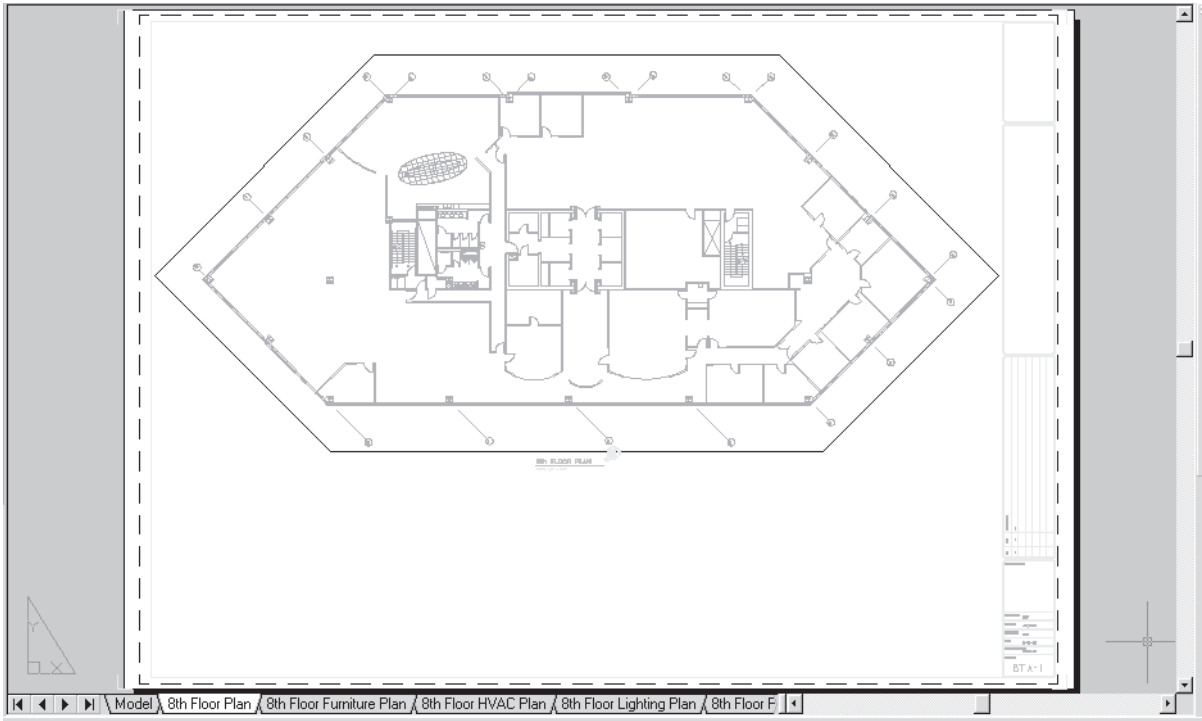
New layouts are created from scratch or imported from other drawings, and can be moved, resized, and deleted. The order in which layouts appear can be changed. As well, each viewport can have a different set of layers visible, and have distinctive plot styles and visual styles. While layouts are always rectangular, viewports can be any shape.

Exploring Layouts

To experience layouts, open the *8th floor.dwg* file from the Companion CD, a floor plan drawing. If necessary, select the **Model** tab. You see the entire model, but without notes, drawing borders, or title blocks.

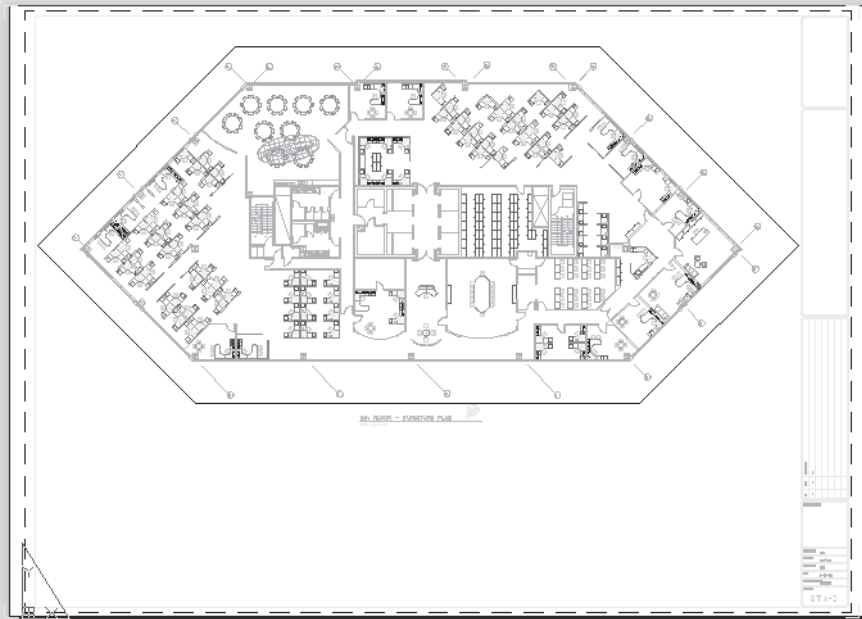


From among the layout tabs, select the **8th Floor Plan** tab by clicking on its name. The view changes dramatically. You see only the floor plan of the model, surrounded by the title block and drawing border. This illustrates how layers can be frozen in layouts to show details selectively.



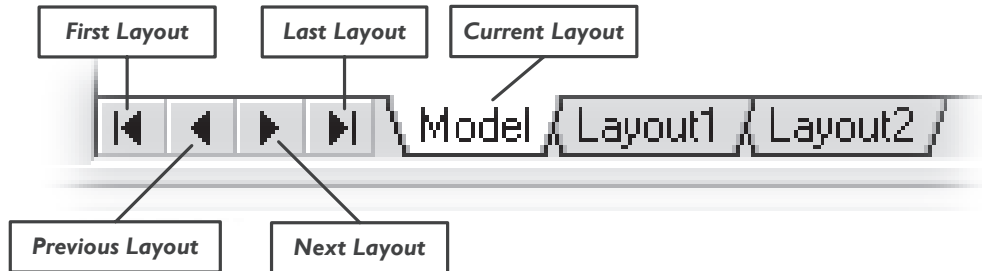
The drawing is shown on a white rectangle, which represents the paper. In this case, the selected paper is “Architectural E1” size, which works out to 42" wide x 30" high. The dashed rectangle represents the printer’s *margin*, the unprintable area along the four edges of the paper.

Select the **8th Furniture Plan** tab. The view changes again. In this layout, layers are turned on to show the floorplan with the furniture — desks, chairs, and so on.



Select the other layout tabs, working your way through the HVAC plan (heating, ventilation, air conditioning), lighting plan, power plan, and plumbing plan. As an alternative to clicking tabs with the cursor, press **CTRL+PGUP** and **CTRL+PGDN** to switch between layouts.

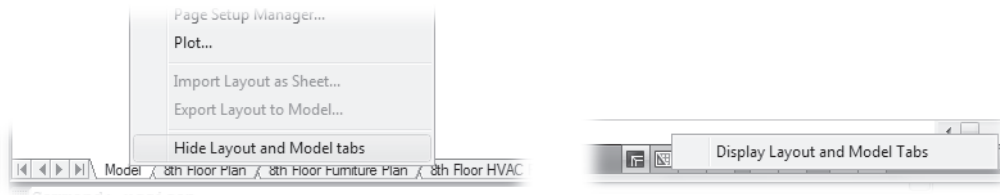
When drawings have many layouts, an additional set of controls becomes useful. These “VCR” buttons take you to the next and previous layouts, as well as to the first and last layouts.



“Hiding” Layout Tabs

Layout tabs can be removed, but I don’t see much point to this option. The tabs share space with the horizontal scroll bar, which I find useful for panning, and so the tabs don’t take up any extra room in AutoCAD’s user interface.

Still, here’s how to do it: right-click any tab. From the shortcut menu, choose **Hide Layout and Model Tabs**. AutoCAD doesn’t hide the tabs, but relocates them as buttons and shortcut menus on the status bar, replacing the MODEL/PAPER button. To move the tabs back to the scroll bar, right-click the model or layout tab, and then choose **Display Layout and Model Tabs**.



 — displays model space.

 — displays the last-accessed layout.

 — displays a list of layout names and preview thumbnail images. Select a thumbnail to go to that layout.



Note To switch between model and paper mode, double-click the layout, as follows:

- Double-click **inside** the viewport to enter model mode; or click the **VsMax** button.
- Double-click **outside** the viewport to return to paper mode.

HISTORY: TILEMODE

Using the **TILEMODE** system variable was the original method of switching between model and *paper space*, as layouts were originally called. With the introduction of layout tabs in AutoCAD 2000, the need for **TILEMODE** as a “command” disappeared.

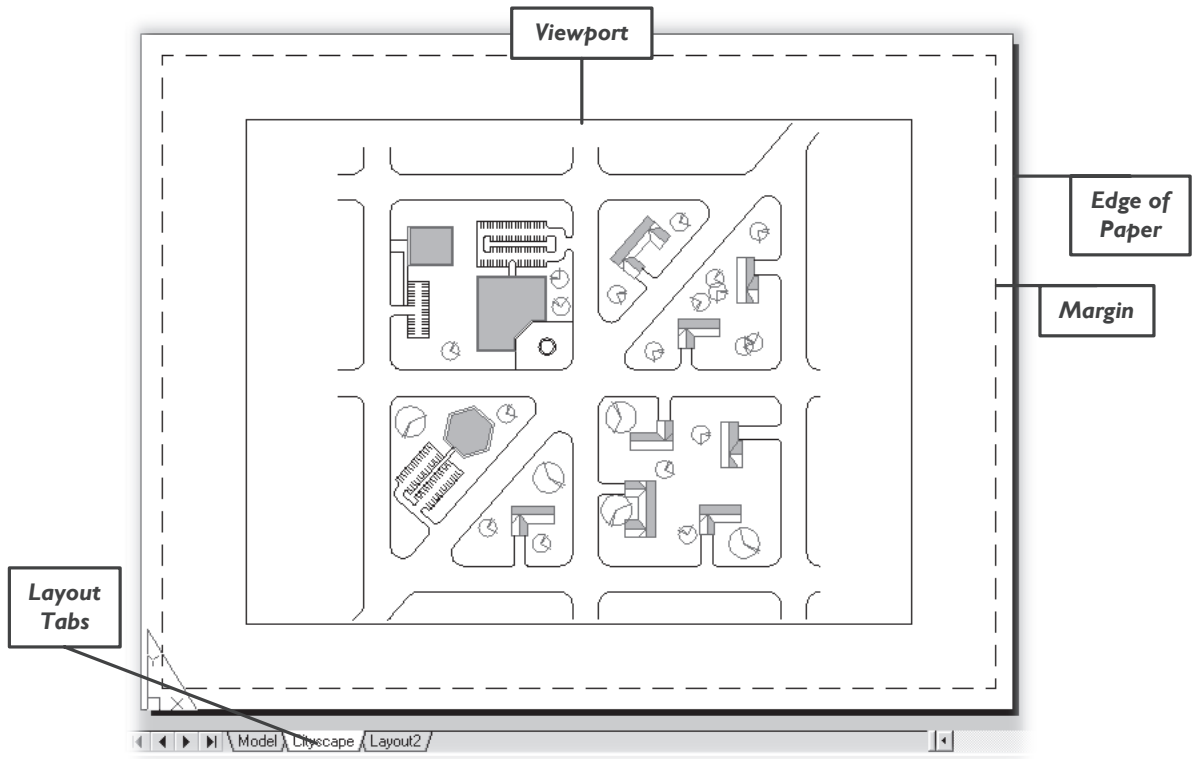
I mention the system variable to point out that when you select model or layout tabs, AutoCAD still executes **TILEMODE 0** or **1** silently in the background.

TUTORIAL: WORKING WITH LAYOUTS

This tutorial takes you through some aspects of working with layouts.

1. From the Companion CD, open the *layout.dwg* file, the drawing of a streetscape.
2. Select the **Layout1** tab by either method:
 - If layout tabs are present, click the one labeled “Layout1.”
 - If layout tabs are not present, click the  layout button.

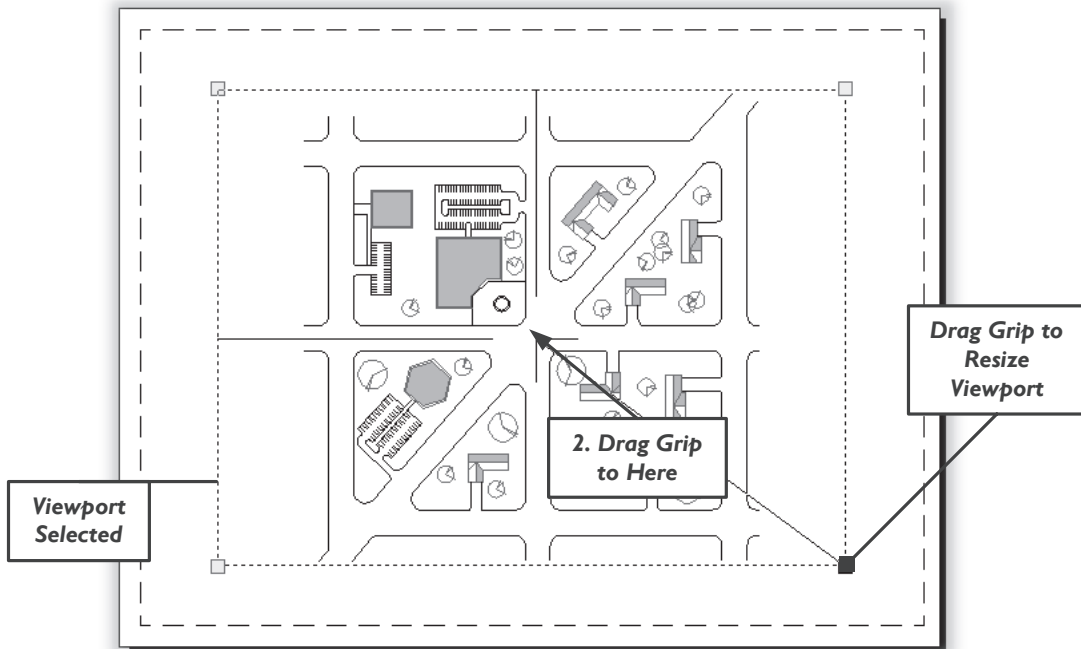
Notice that the cityscape drawing appears on a sheet of paper with two rectangles. The inner rectangle is the viewport, while the dashed rectangle is the printer margin.



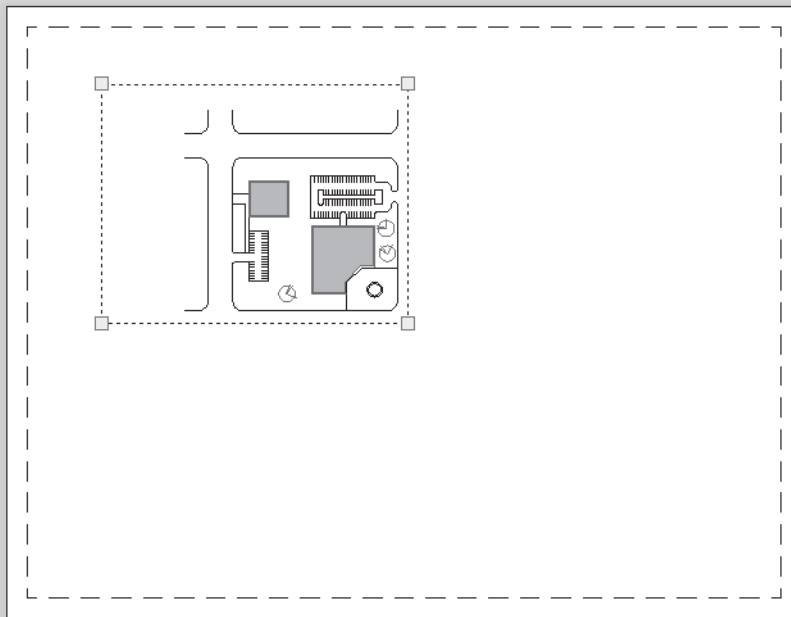
In layout mode, you work only with the “paper,” not the model. “Working with the paper” includes drawing on the paper or page, manipulating viewports and their content, and plotting. You’ll find that you cannot touch the model at all.

1. Try selecting the model. Notice that you can’t, because the layout is in “paper” mode. (On the status bar, the indicator reads PAPER.)

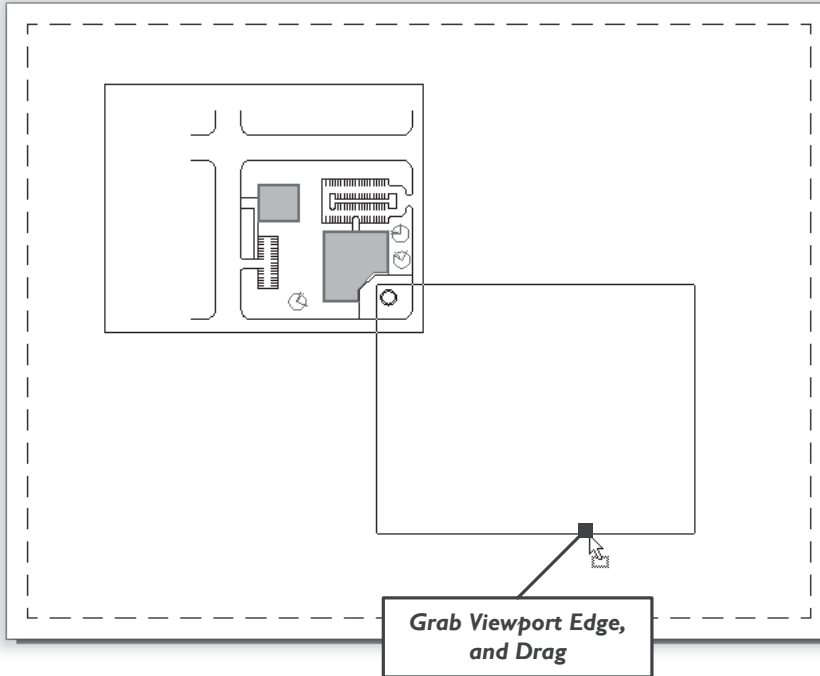
- Click the viewport border. Notice that you can select it, unlike the model.



- Grab a grip, and then make the viewport smaller by dragging. Notice what happens to the model: it has become “cropped” — part of the model is cut off visually. This illustrates how you can selectively show details of the model in layouts. Layouts need not show the entire model.

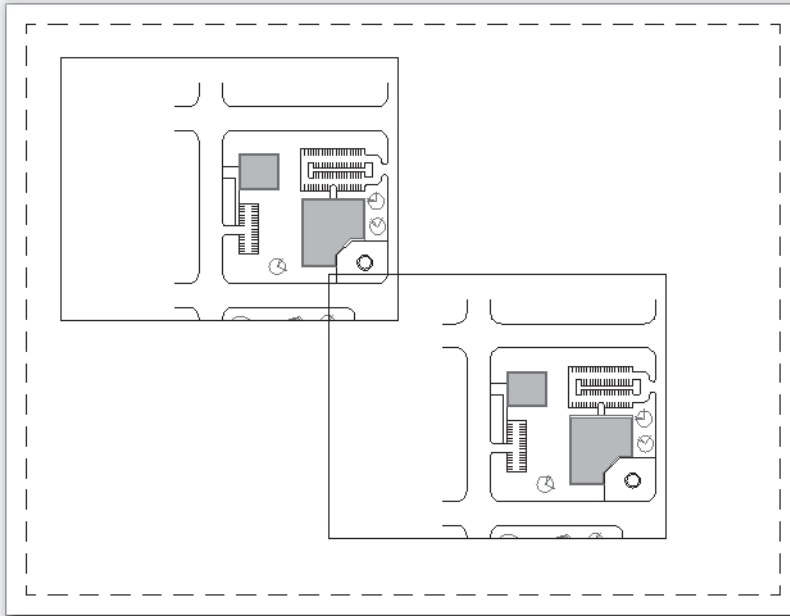


4. You can move viewports: grab an edge (not a handle), and then drag the viewport to a new position. Notice that the cropped model view moves with the viewport.



You can apply all of the usual grip editing commands to viewports: stretch, move, copy, scale (resize), rotate, and mirror.

5. Use the **COPY** command, for example, to copy viewports. Notice that the copy of the viewport contains exactly the same model view as the original. And, as the figure below illustrates, viewports can overlap.



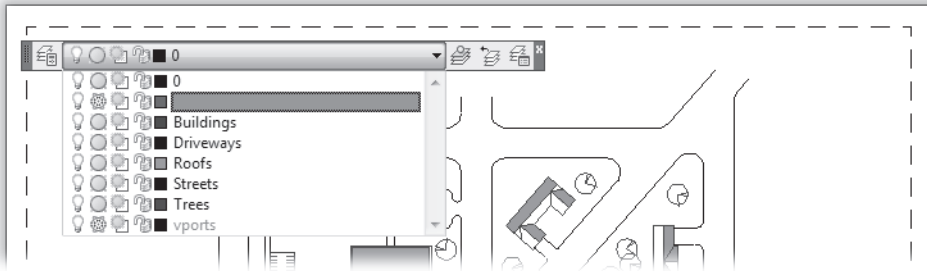
You can change the color of viewport borders, or make them invisible by placing them on a layer of their own, and then change the color or freeze the layer. The rest of the model is unaffected. (You cannot apply linetypes or lineweights to viewports borders.) To see viewport borders in the drawing, but not plot them, change the layer to be non-plotting.

6. The **ERASE** command deletes viewports. Other editing commands, however, do not work with viewports, including **OFFSET**, **TRIM**, **EXPLODE**, and **FILLET**.

Hiding Viewport Borders

If you don't want the viewport borders to appear in drawings, use this trick to hide them: assign viewports their own layer, and then freeze the layer. This makes the viewport borders invisible, but the content remains visible.

1. With the **LAYER** command, create a new layer named "VPorts."
2. Select the viewport borders, and from the Layers toolbar or ribbon panel, click the **Layer Control** droplist to change the layer to "VPorts."



EDITING VIEWPORTS

You can make many kinds of changes to viewports using the Properties palette. Some of the properties, however, don't work. For example, it would appear that Properties lets you apply linetypes and lineweights to the viewport border, but changing these properties has no effect.

Viewport Border Properties

The following properties can be applied to viewport borders:

- Color** — changes the color of the viewport border.
- Layer** — specifies the border's layer name.
- Plot Style** — specifies the plot style, if enabled.
- Hyperlink** — activates a link when the border is clicked.
- Geometry** — specifies the center coordinates, and the size.

Viewport Content Properties

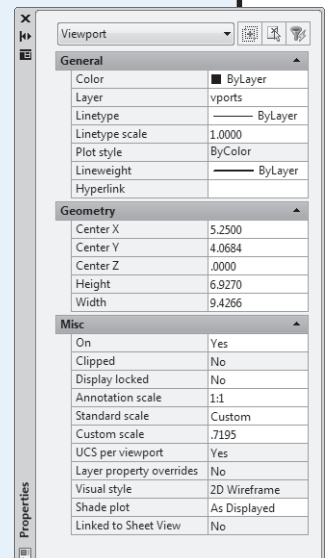
The following properties apply to the content of viewports:

- On** — toggles the display of the view in the viewport.
- Display Locked** — prevents view changes, such as zooms and pans.
- Standard Scale** — scales the view using factors specified by **SCALELISTEDIT**.
- Custom Scale** — specifies other scale factors.
- Shade Plot** — specifies shading or rendering mode during plotting.

Other Viewport Commands

These properties are controlled by other commands:

- Clipped** — accessed through the **MVIEW** and **-VPORTS** commands.
- UCS Per Viewport** — accessed through the **UCS** command.
- Visual Style** — accessed through the **VSCURRENT** system variable; can't set in paper space.
- Linked to Sheet View** — accessed through the **SHEETSET** command.



3. Freeze layer “VPorts.” Notice that the viewport border disappears, but that the model view remains.
4. Thaw layer “VPorts” to bring back the viewport borders.

Switching from Paper to Model Mode

I lied. There are three ways to view drawings: (1) in the Model tab, (2) in a layout tab, and (3) in model mode of a layout tab. You can edit the model inside the viewport.

Until now, you have been manipulating viewports. Let’s work with the model inside the viewports.

1. On the status bar, click **PAPER**.

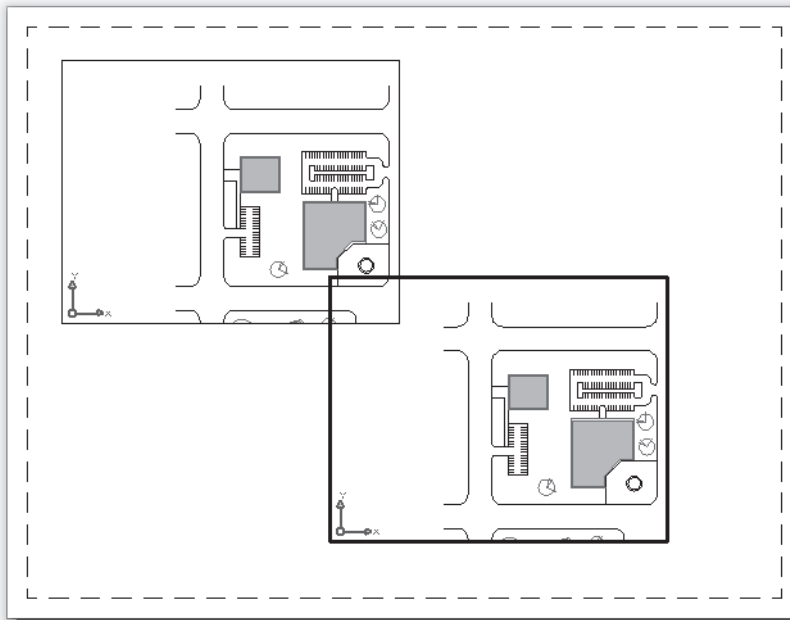


Notice that the button changes its name to **MODEL**. This means that you are now working with the model instead of the layout (paper) — yet the drawing is still in layout mode.



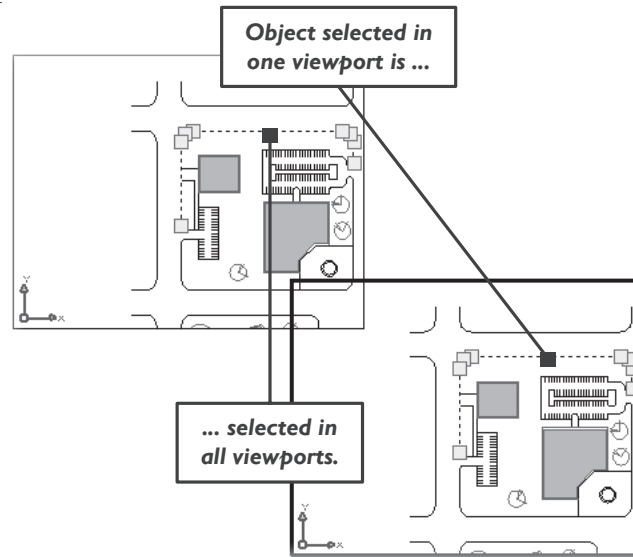
(If you cannot see words **PAPER** or **MODEL** on the status bar, then double-click inside the viewport to enter model space.)

2. Either way, notice that one viewport has a heavy border. This is the *active* viewport, the one in which you can manipulate the model.



To make another viewport active, simply select it. Alternatively, you can press **CTRL+R** to cycle through viewports.

3. Select an object in the model. Notice that the grips and highlighting appear in both viewports. This confirms that you are seeing two images of the same model.



SCALING MODELS IN VIEWPORTS

You can show models at a different scale in every viewport. As well, you can hide and show layers, and override layer properties, such as color and linetype, and toggle the grid — all on a per-viewport basis. In this section, we look at the important topic of scaling models inside viewports.

VP Scale

The easiest way to scale models in viewports is to use the VP Scale droplist found on the status bar. (See illustration at right.)

1. Select a viewport.
2. Click the **VP Scale** list, and then choose a scale factor.

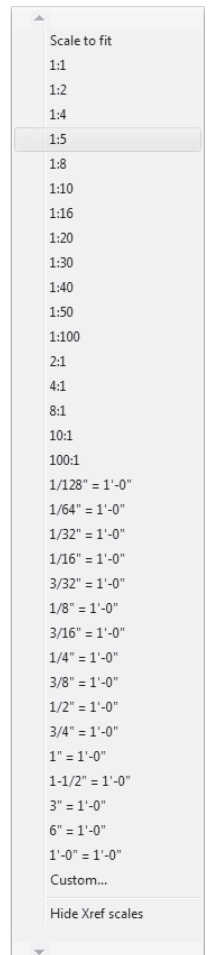
Notice that the model changes its size inside the viewport. You can choose a different scale factor for each viewport — or use the same one for all.

Not sure which scale factor to choose? Read ahead to the section on the **Zoom XP** command.

(This droplist does not work if the viewport is locked, either through the Properties palette's **Display Locked** or the **MVIEW** and **-VPORIS** commands' **Lock** options.)

The scale factors listed by the droplist can be edited with the **SCALELISTEDIT** command. This command lets you add and remove scale factors; you can access it through the **Custom** item on the VP Scale droplist.

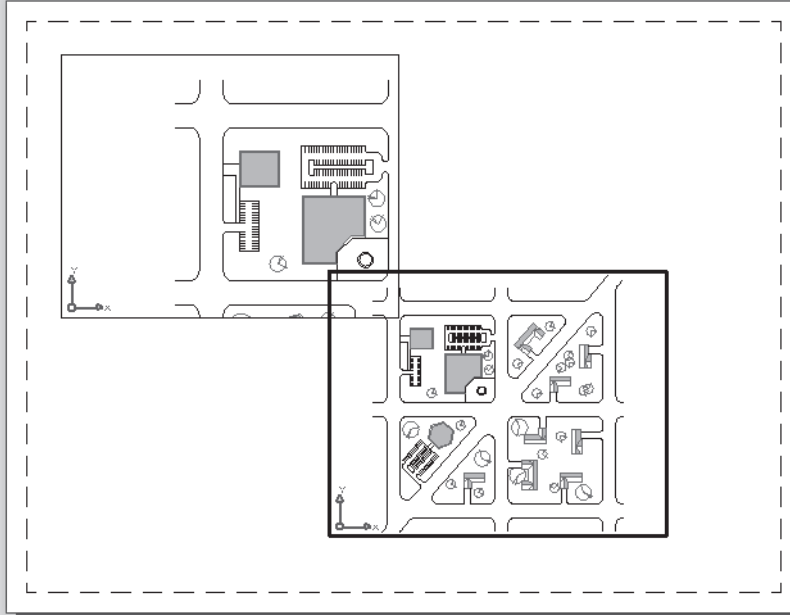
In addition, the **HIDEXREFSCALE** system variable hides scale factors imported from externally-referenced drawings. See Chapter 20, "Plotting Drawings," for more on both of these.



Zoom XP

AutoCAD's two primary visibility commands are **ZOOM** and **LAYER**. In model mode, AutoCAD commands apply to the active viewport only; all other viewports are ignored.

1. The **ZOOM** command changes the size of the model in each viewport. Enter the **ZOOM** command, and then use the **Extents** option. You should see the entire model in the active viewport.



2. The **ZOOM** command's **XP** option is meant for use in layout viewports (short for "multiple in paper"). This option scales the model view relative to the layout. This is important: *You use **ZOOM XP** to scale models in layout viewports.*

Out of so many scale factors, which one is appropriate for this drawing? City plans have typically large scale factors, such as 1:1000, 1:2000, 1:5000, and so on. This model is approximately 470 feet across, while the viewport is about 4.7 inches across. Doing the math shows that the nearest standard scale factor is 1:2000:

$$(470' \times 12 \text{ inches/foot}) \div 4.7" = 1200$$

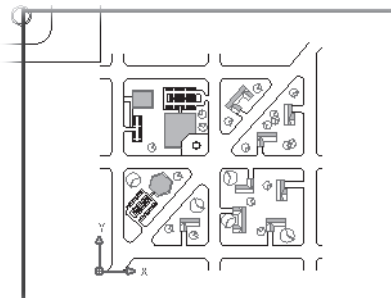
... so we select the next-highest standard scale factor.

3. Apply the scale factor of 1:2000 by using it with the XP suffix, as follows:

Command: **zoom**

Specify corner of window, enter a scale factor (nX or nXP), or

[All/Center/Dynamic/Extents/Previous/Scale/Window/Object] <real time>: **1/2000xp**



Notice that the layout now has two views of the model: a large one and a small one. This illustrates that viewports can have independent views of models.



Notes Use the **PAN** command to position the model inside viewports.

You can lock viewports to prevent anyone from changing the view of the model, such as through zooming or panning: (1) In paper mode, select the viewport border, or (2) In the Properties palette, change the **Display Locked** property to **Yes**.

To enter model space without messing up the viewport scale, click the **Maximize Viewport** button.

The viewport scale factor can also be set with the Properties palette. It has the advantage of providing access to the QuickCalc calculator, which is useful when you need to work out the scale factor.

SELECTIVELY DISPLAYING DETAILS IN VIEWPORTS

Earlier in this chapter, you saw how it was possible to make the viewport border invisible by freezing its layer. The **LAYER** command also allows you to change what is displayed *inside* the viewports on a per-viewport basis:

Freeze layers — hides parts of the drawing; thaw the layers to make the parts visible again.

Override layer properties — displays parts of drawings with different colors, linetypes, and so on.

The “ByLayer” properties that can be overridden in each viewport are color, linetype, linewidth, and plot style. Whether freezing or overriding, the process is the same:

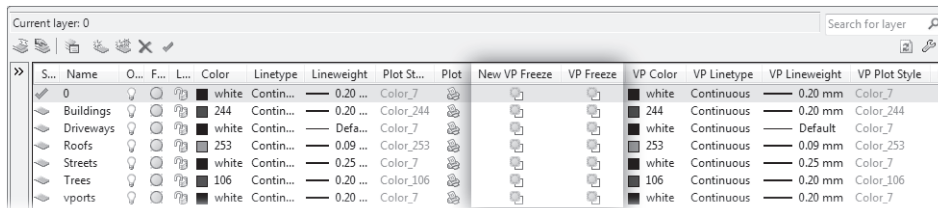
1. Select a viewport.
2. Use the **LAYER** command to freeze layers or override properties.
3. Repeat for other viewports.

Let’s look at this in some detail.

Freezing Layers

The **LAYER** command freezes layers in the current viewport. This is done with its **Current VP Freeze** column.

1. Ensure the *layout.dwg* drawing is open. Select one of the viewports.
2. With the **LAYER** command, open the Layer Properties Manager palette.



Towards the center are two columns you have not yet used:

- **Current VP Freeze** column freezes the selected layers in the current viewport (a.k.a. “VP”) — as soon as you make the change in the palette (or click **Apply** in the dialog box).
- **New VP Freeze** column freezes the selected layers — whenever a new viewport is created.

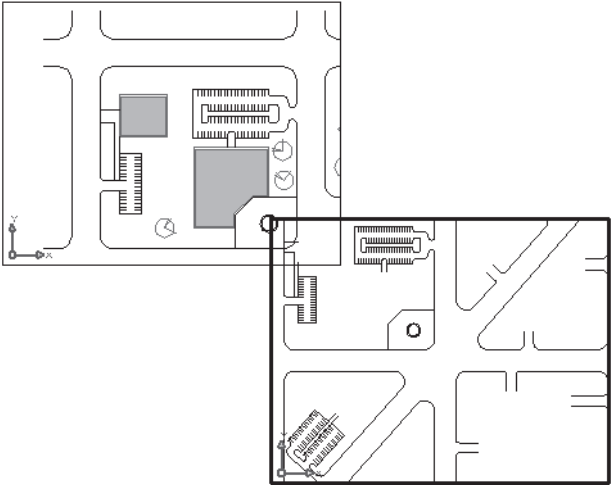
These two options have no effect in model tab and in unselected layouts and viewports.

3. Hold down the **CTRL** key, and then select names of the **Buildings**, **Roofs**, and **Trees** layers, as illustrated below.

S...	Name	O...	F...	L...	Color	Linetype	Lineweight	Plot St...	Plot	New VP Freeze	VP Freeze	VP Color	VP Linetype	VP Lineweight	VP Plot Style	D
✓	0				white	Contin...	0.20 ...	Color_7				white	Continuous	0.20 mm	Color_7	
	Buildings				244	Contin...	0.20 ...	Color_7				244	Continuous	0.20 mm	Color_7	
	Driveways				white	Contin...	Defa...	Color_7				white	Continuous	Default	Color_7	
	Roofs				253	Contin...	0.09 ...	Color_7				253	Continuous	0.09 mm	Color_7	
	Streets				white	Contin...	0.25 ...	Color_7				white	Continuous	0.25 mm	Color_7	
	Trees				106	Contin...	0.20 ...	Color_7				106	Continuous	0.20 mm	Color_7	
	vports				white	Contin...	0.20 ...	Color_7				white	Continuous	0.20 mm	Color_7	

Click icon in
Current VP Layer column

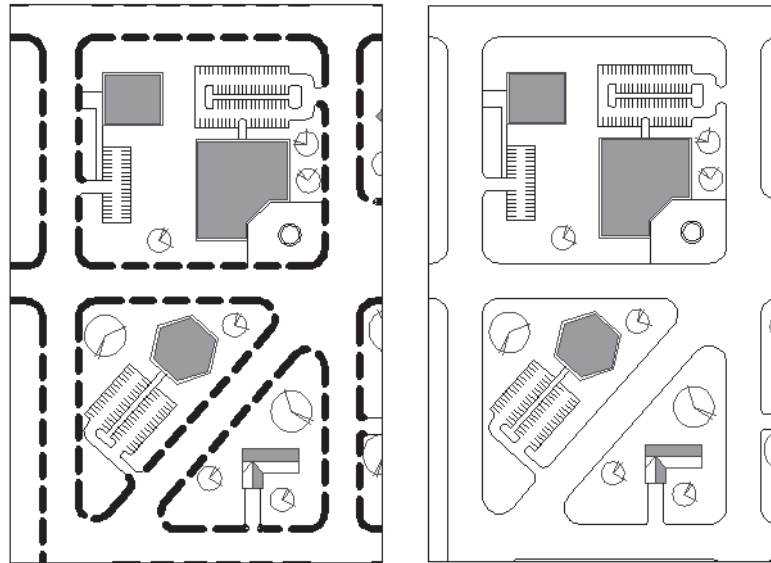
4. Let go of the **CTRL** key, and then click a highlighted icon in the **Current VP Freeze** column. Notice that all selected suns turn to snowflakes, indicating the layers will be frozen (hidden from view).
5. Choose the **OK** button. Notice that the buildings, roofs, and trees disappear from the viewport. The other viewport, however, is unaffected. This shows that viewports can have their layers frozen independently of each other.



Overriding Layer Properties

You can override layer properties with commands like **COLOR** and **LINETYPE**, but these overrides affect all viewports equally. The **LAYER** command can override properties in each viewport.

For example, you might have some layers colored light gray in one viewport but black in others. Some viewports might show layers with the Hidden linetype, while others show the Continuous linetype, as illustrated below.



Left: Streets in hidden linetype and heavy lineweight overrides.

Right: Streets displayed normally (ByLayer).

The **LAYER** command overrides properties with the VP Color, VP Linetype, VP Lineweight, and VP Plot Style columns.

1. Continuing with the *layout.dwg* drawing, double-click inside a viewport to enter model space.

Use the **U** command to undo the layer changes from the previous tutorial.

2. Open the Layer Properties Manager dialog box with the **LAYER** command.

S...	Name	O...	F...	L...	Color	Linetype	Lineweight	Plot St...	Plot	New VP Freeze	VP Freeze	VP Color	VP Linetype	VP Lineweight	VP Plot Style
✓	0				white	Contin...	0.20 ...					white	Continuous	0.20 mm	
	Buildings				244	Contin...	0.20 ...	Color_244				244	Continuous	0.20 mm	Color_244
	Driveways				white	Contin...	Defa...	Color_7				white	Continuous	Default	Color_7
	Roofs				253	Contin...	0.09 ...	Color_253				253	Continuous	0.09 mm	Color_253
	Streets				white	Contin...	0.25 ...	Color_7				white	Continuous	0.25 mm	Color_7
	Trees				106	Contin...	0.20 ...	Color_106				106	Continuous	0.20 mm	Color_106
	vports				white	Contin...	0.20 ...	Color_7				white	Continuous	0.20 mm	Color_7

Towards the right are the four columns that we are interested in:

- **VP Color** column overrides the ByLayer color in the current viewport.
- **VP Linetype** column overrides the ByLayer linetype.
- **VP Lineweight** column overrides the ByLayer lineweight.
- **VP Plot Style** column overrides the ByLayer plot style, if plot styles are enabled.

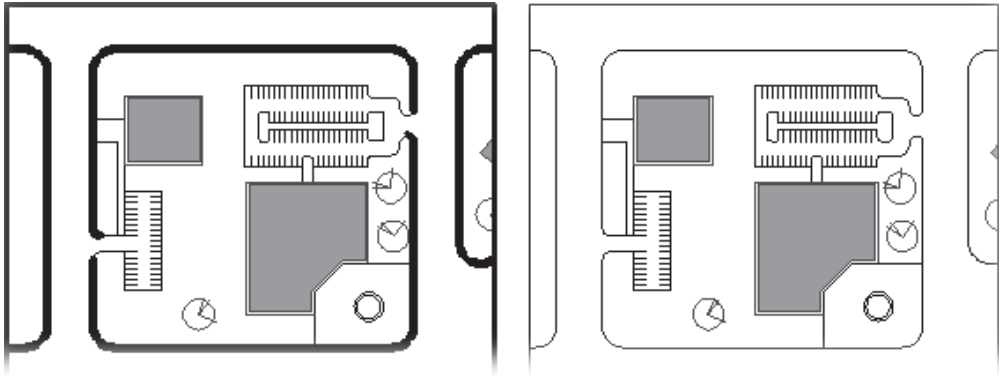
3. Select names of the **Streets** layer, as illustrated below.

4. In the VP Lineweight column, select 2.11 mm.

S...	Name	O...	F...	L...	Color	Linetype	Lineweight	Plot St...	Plot	New VP Freeze	VP Freeze	VP Color	VP Linetype	VP Lineweight	VP Plot Style
✓	0				white	Contin...	0.20 ...	Color_7				white	Continuous	0.20 mm	Color_7
	Buildings				244	Contin...	0.20 ...	Color_244				244	Continuous	0.20 mm	Color_244
	Driveways				white	Contin...	Defa...	Color_7				white	Continuous	Default	Color_7
	Roofs				253	Contin...	0.09 ...	Color_253				253	Continuous	0.09 mm	Color_253
	Streets				white	Contin...	0.25 ...	Color_7				white	Continuous	2.11 mm	Color_7
	Trees				106	Contin...	0.20 ...	Color_106				106	Continuous	0.20 mm	Color_106
	vports				white	Contin...	0.20 ...	Color_7				white	Continuous	0.20 mm	Color_7

Change value in
VP Lineweight column

- 5. Notice that the streets are displayed with heavy lines in the current viewport; the other viewport is unaffected by the change.



Left: Streets in heavy lineweight override.
Right: Streets displayed normally (ByLayer).



Note The **VPLAYEROVERRIDES** system variable reports whether any viewports contain layer property overrides. When “1,” at least one does; when “0,” none do. (This sysvar is read-only).

The same information is provided by the Properties palette. Select a viewport border, and then check whether the value of **Layer Property Overrides** is Yes or No. Like the sysvar, this setting is read-only.



VpOverrideMode — Toggling Properties Overrides

You can toggle property overrides with the **VPOVERRIDE** system variable. It has two settings:

- 1 — overrides are displayed and plotted.
- 0 — overrides are neither displayed nor plotted.

This sysvar is handy for temporarily turning off overrides. To turn them off “permanently,” see the next section.

Reverting Properties to ByLayer

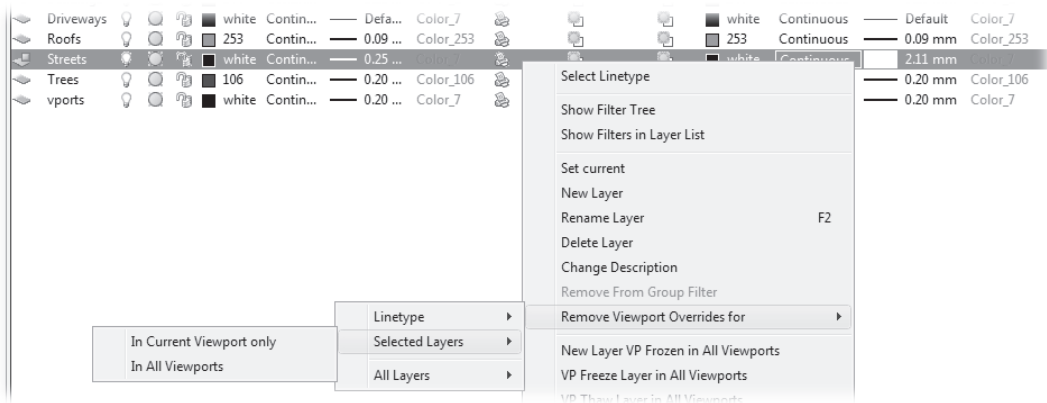
After you’ve overridden the layer properties of several viewports, you may want to change them back to “normal” (ByLayer). Autodesk provides three (!) ways to do this: through the **LAYER**, **-VPOR**, and **MVIEW** commands.

The **LAYER** command’s palette indicates overridden properties by highlighting them in cyan (light blue). In addition, a different icon  appears in the Status column; pausing the cursor over the icon displays a yellow tooltip that reports the override value(s).



To remove overrides, follow these steps:

1. In the Layer dialog box, right-click any layer name.
2. From the shortcut menu, select **Remove Viewport Overrides For**.



Notice the options:

- **Selected Layers** — removes overrides from selected layers only.
 - **All Layers** — removes overlays from all layers.
 - For both options, overrides can be removed from the current viewport or all viewports.
3. Select an option. Notice that the overrides and cyan highlights disappear.

The MVIEW and -VPORTS commands operate identically, curiously enough. Both have the LAYER option that removes the overrides. Here are the prompts for MVIEW:

Command: **mview**

Specify corner of viewport or [ON/OFF/Fit/.../LAYER/2/3/4] <Fit>: **la**

Reset viewport layer property overrides back to global properties? [Yes/No]: **y**

Select objects: *(Select one or more viewports.)*

Select objects: *(Press ENTER to exit object selection.)*

Notice that the overrides are removed.

INSERTING TITLE BLOCKS

Layout mode is where you place scale-dependent objects, such as drawing notes and title blocks. In this tutorial, you add a title block and border to a drawing.

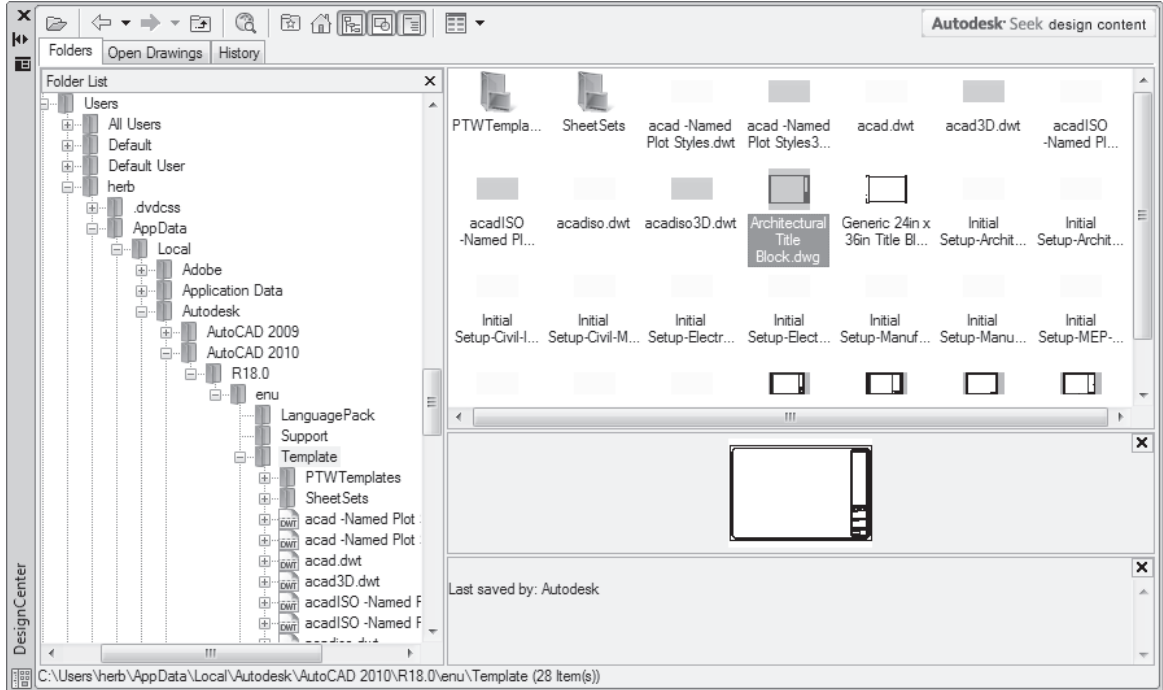
1. Ensure that Layout1 is in PAPER mode (no viewport with a heavy outline).
2. Create a new layer called "Titleblock," and then make it current.
3. Save the drawing with the **QSAVE** command.
4. Open Design Center (press **CTRL+2**).
5. In the Folder List, go to the folder holding the template drawings (not in the `\template` folder, which is empty):

`C:\Users\<login name>\AppData\Local\Autodesk\AutoCAD 2010\R18.0\enu\Template`

Replace `<login name>` with your Windows login name, such as "administrator."



Note The folder is inconveniently hidden by Windows, and so it might not show up in DesignCenter. Here is one way to access it: enter the **ADCNAVIGATE** command and then provide the folder path listed above. DesignCenter immediately displays the contents of the *template* folder.



6. Drag the *Architectural Title Block.dwg* file from DesignCenter into the drawing.
7. When AutoCAD prompts you for the insertion point, pick the lower-left corner of the margin.
 Command: `_INSERT` Enter block name or [?]: `<A$C27FF03C9>`: "ANSI A title block.dwg"
 Specify insertion point or [Scale/X/Y/Z/Rotate/PScale/PX/PY/PZ/PROtate]: *(Pick the lower left corner of the margin.)*
8. Press **ENTER** at each of the remaining prompts.
 Enter X scale factor, specify opposite corner, or [Corner/XYZ] `<1>`: *(Press ENTER to keep scale factor at 1.0.)*
 Enter Y scale factor `<use X scale factor>`: *(Press ENTER to keep scale factor at 1.0.)*
 Specify rotation angle `<0>`: *(Press ENTER to keep angle at 1.0.)*
9. You may need to move the viewports so that they do not interfere with the title block or border.

USING TILEMODE

At the start of this chapter, I mentioned the **TILEMODE** system variable, which has been replaced by the layout tabs. For completeness, here is how to use it:

Command: **tilemode**

Enter new value for TILEMODE `<0>`: **1**

Entering **1** returns to the model tab, while entering **0** returns to the last active layout tab.



The **VPMAX** command maximizes the size of the current viewport (short for “viewport maximize”). This command is useful for increasing the size of small viewports. Conversely, the **VPMIN** command returns the viewport to its normal size. (As an alternative, you can double-click viewport borders to maximize and minimize them.)

Using this pair of commands is faster than switching back and forth between paper and model space, because AutoCAD automatically switches to model space at the same viewing scale as the layout. Another advantage is that these two commands do not affect model/paper space zoom scales.

The **VPMAX** and **VPMIN** commands work in layouts only, and not in Model tab. Drawings cannot be plotted or published while the viewport is maximized.



Left: Maximize Viewport button.

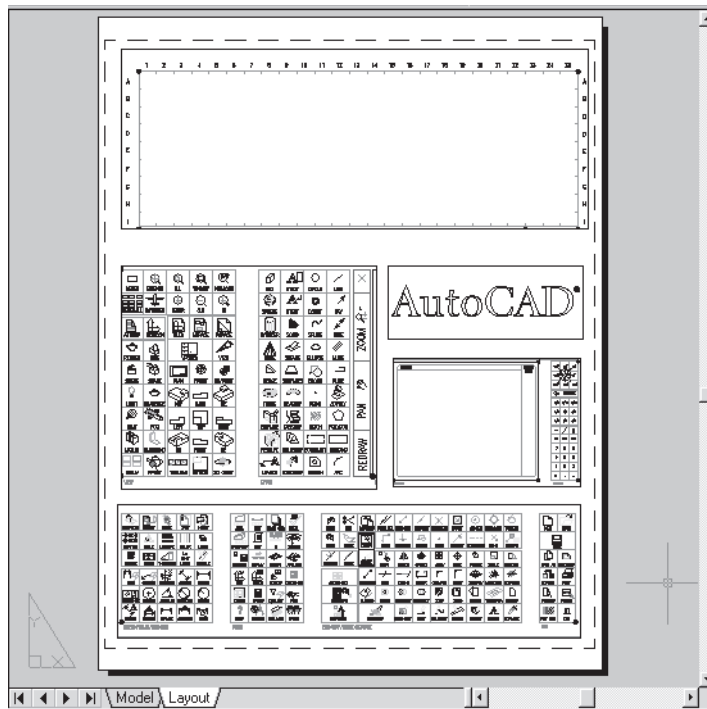
Right: Minimize Viewport button surrounded by the Previous and Next buttons.

When the layout has more than one viewport, two arrow buttons appear, as illustrated above at right. These buttons allow you to move to the next and previous viewport, in the order in which they were created. To return the viewport to its normal size, click the **Minimize Viewport** button.

TUTORIAL: MAXIMIZING AND MINIMIZING VIEWPORTS

1. For this tutorial, open the *vpmax.dwg* file from the CD.

Click the **Layout** tab to ensure the drawing displays the viewports in layout mode.

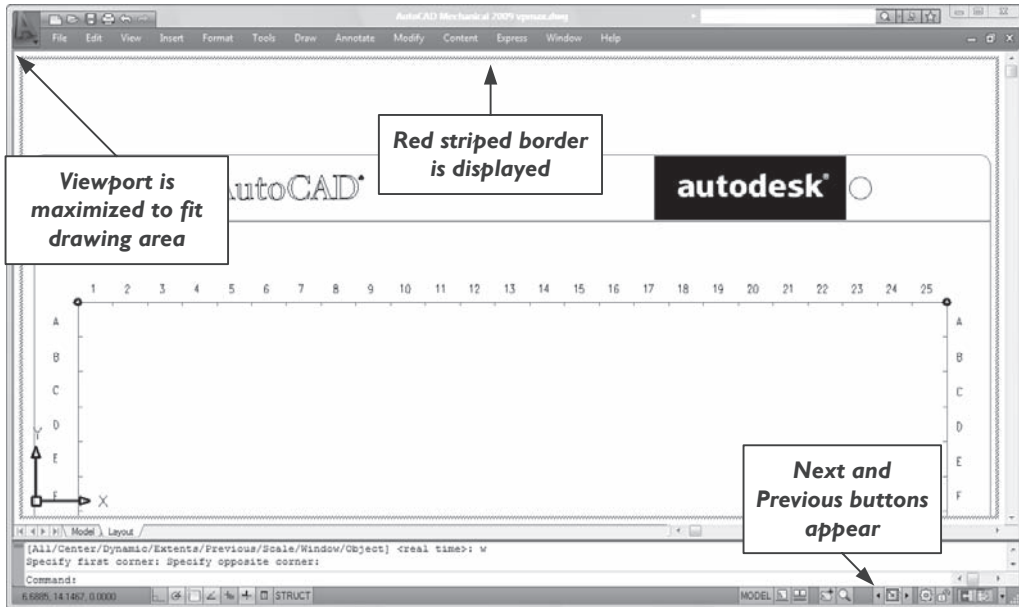


2. To maximize viewports, start the **VPMAX** command:
 - In paper space, double-click a viewport border.
 - In paper space, select a viewport. Right-click, and then select **Maximize Viewport** from the shortcut menu.

- On the status bar, click the **Maximize Viewport** button.
- At the 'Command:' prompt, enter the **vpmax** command.
Command: **vpmax** (Press ENTER.)

3. If the layout contains more than one viewport, AutoCAD asks you to select one of them:
Select a viewport to maximize: (Select a viewport.)

Notice that AutoCAD does the things illustrated in the figure below:



- Viewport is made as large as the drawing area.
- Viewport has a red striped border.
- Previous and Next buttons appear on the status bar, if the layout contains more than one viewport. And the Maximize Viewport button turns into the Minimize Viewport button.
- Space changes from paper to model.



4. To see other viewports in their maximized state, click the **Maximize Previous Viewport** or **Maximize Next Viewport** button. Notice that AutoCAD displays the next viewport every time you click the button.
5. When done editing, minimize the viewport using one of these methods:
 - Double-click the red striped border.
 - Right-click anywhere inside the viewport, and then select the **Minimize Viewport** option from the shortcut menu.
 - On the status bar, click the **Minimize Viewport** button.
 - At the 'Command:' prompt, enter the **vpmin** command.

Command: **vpmin** (Press ENTER.)

AutoCAD returns the viewport to its natural size.



Note Here's how to maximize a viewport but remain in paper space: select a viewport, and then use the **zoom** command with its **Object** option. AutoCAD maximizes the viewport.



The **QVLAYOUT** command displays a filmstrip-like interface for switching layouts (short for “quick view layouts”).

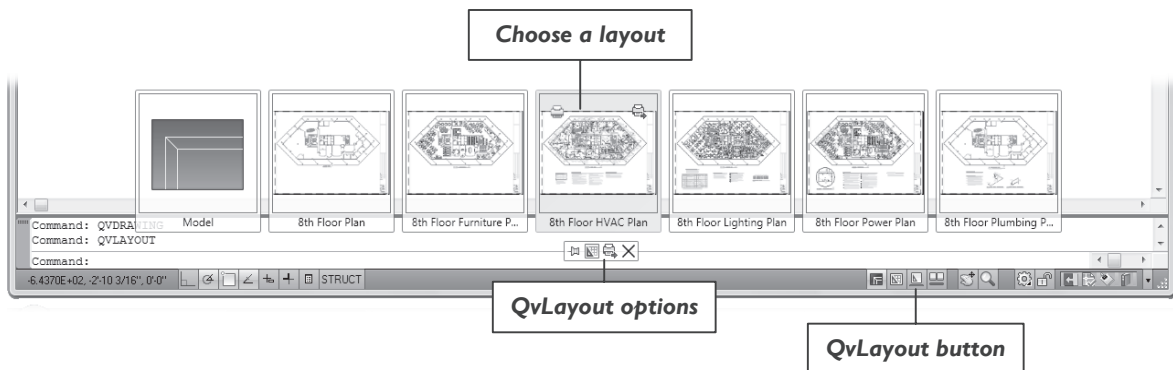
Autodesk added this interface to AutoCAD 2009 as a “quicker” and more visual way of accessing layouts. Frankly, I think that layout tabs are better, but the decision is up to you. The best way to understand quick views is to experience it.

TUTORIAL: SWITCHING LAYOUTS THROUGH QUICK VIEWS

1. For this tutorial, open a drawing from the Companion CD that has many layouts: *8th floor.dwg*.
2. To switch to another layout, start quick views through one of the following methods:
 - On the status bar, click the  Quick View Layouts button.
 - At the ‘Command:’ prompt, enter the **qvlayout** command.

Command: **qvlayout**

Notice the filmstrip of thumbnails.



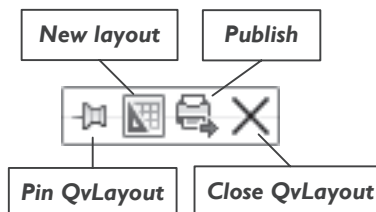
3. To switch to a layout, click on its thumbnail image.



Notes If there are quick views beyond the width of your monitor, move the mouse past the edge of the image strip to see additional thumbnails.

To change the size of the thumbnails, hold down the **CTRL** key while rotating the mouse’s roller button back and forth. This makes the images larger and smaller.

This user interface has a number of features. The small toolbar-like rectangle contains these options:



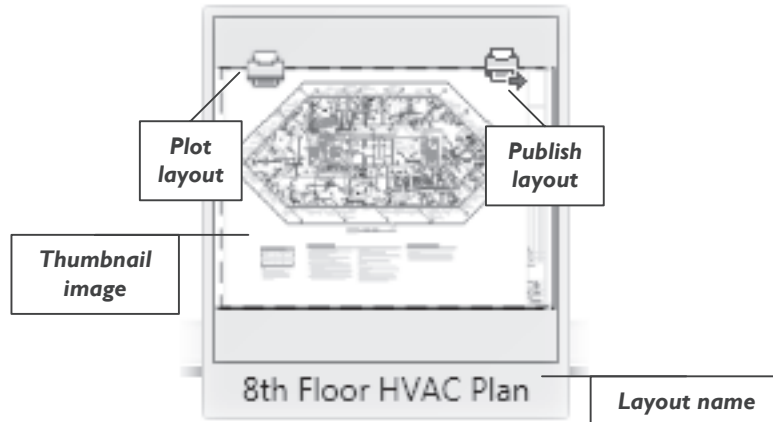
Pin — “pins” the quick view layout, keeping it open until you click the **x** button; when unpinned, the quick view closes as soon as you pick any point outside of it.

New — creates new layouts; see the **LAYOUT** command later in this chapter.

Publish — publishes the layout; see the **PUBLISH** command in the next chapter.

Close — closes the quick view interface; alternatively, use the **QVLAYOUTCLOSE** command.

When the cursor passes over a thumbnail image, the following controls become available:



Plot — plots the layout; see **PLOT** command in the next chapter.

Publish — publishes the layout.



'SPACETRANS

The **SPACETRANS** command converts distances between model space units and paper space units (short for “space translation”).

This command automatically determines the height of text being placed in model mode of a layout, but should be scaled appropriately for paper mode. The command is meant to be used transparently, during other commands when AutoCAD asks for heights, lengths, or distances. However, **SPACETRANS** only operates transparently after it has been run normally at least once.

The command may only be used in layouts — either in paper or model mode; attempting to use it in model space results in this complaint:

**** Command not allowed in Model Tab ****

(You will find it easier to use annotative scaling to scale text than this command.)

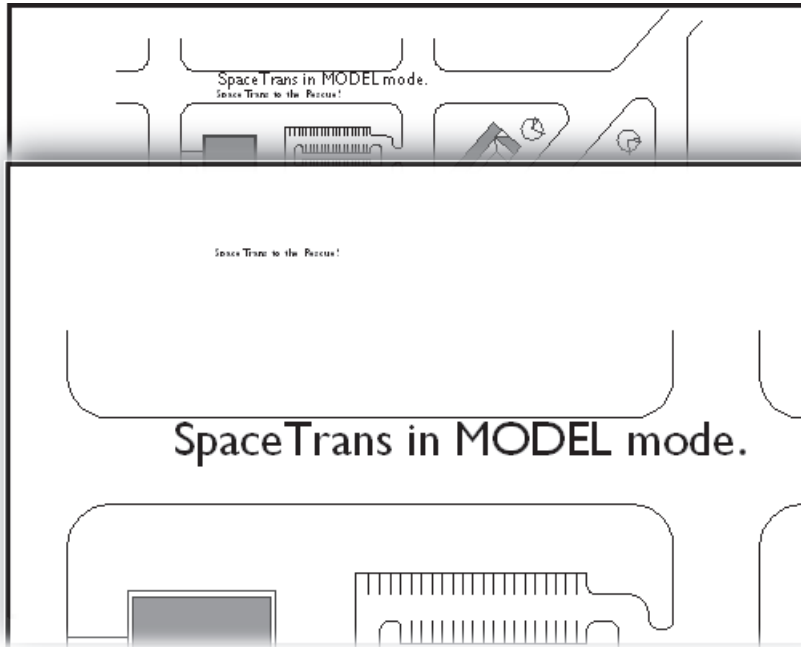
TUTORIAL: DETERMINING TEXT HEIGHT IN LAYOUTS

1. To specify that the height of text in model space match a given height in a layout, ensure AutoCAD is displaying model space through the viewport in the layout, and then enter the following:
 Command: **text**
 Specify start point of text or [Justify/Style]: *(Pick a point.)*
 Specify height <0.2>: **'spacetrans**
2. You can enter a fraction, and AutoCAD calculates it. For example, 1/8" (0.125) is a typical height for text in drawings.
 >>Specify paper space distance <1.000>: **1/8**

Press **ENTER**, and AutoCAD resumes the **TEXT** command, displaying the calculated height (0.125 x viewport scale factor):

Resuming TEXT command.

Specify height <0.2000>: 0.075783521943615



3. Continue with the command's other prompts:

Specify rotation angle of text <0>: (Press **ENTER**.)

Enter text: **SpaceTrans to the Rescue!** (Press **ENTER**.)

Enter text: (Press **ENTER**.)

The text is placed in the model at a size of 0.075 — instead of 0.125.



Notes Use **SPACETRANS** to create notes in model space that appear at the correct height in the layout, and are plotted at the correct size.

Use it together with the **SCALETEXT** command to change existing text. Start the **SCALETEXT** command, and then enter '**SPACETRANS**' to specify the new height. AutoCAD scales the text correctly.

SPACETRANS cannot be used with the **MTEXT** command. Instead use it as a calculator at the 'Command:' prompt to find the scaled text height, and then enter the height manually in the mtext editor:

Command: **spacetrans**

Specify model space distance <0.1250>: **1.0**

0.663080708963838

When the viewport scale changes, the text also changes size, depending on the mode in which it was placed:

When text is placed in **PAPER** mode

- It is not visible in model space.
- It does not change size when viewport scale changes.
- It issues the following prompts (note the word "model"):

Select a viewport: *(Select a viewport border.)*

Specify model space distance <1.0>: *(Enter a value.)*

When the text is placed in MODEL mode

- It is visible in model space.
- Its apparent size changes as seen from the layout.
- It issues the following single prompt (note the word “paper”):

Specify paper space distance <1.000>: *(Enter a distance.)*

PSLTSCALE

The **PSLTSCALE** system variable matches linetype scaling to viewport scaling, so that linetypes drawn in the model appear the correct size (short for “paper space line type scale”).

Command: **psltscale**

Enter new value for PSLTSCALE <1>: **0**

When set to **1** (the default), the viewport scale factor (set by **ZOOM Xp** or the viewport properties) controls linetype scaling. The lengths of dashes and gaps are based on paper space units — for objects drawn in model space and in layouts. The advantage is that viewports can have different zoom levels (scale factors), yet the linetypes look the same.

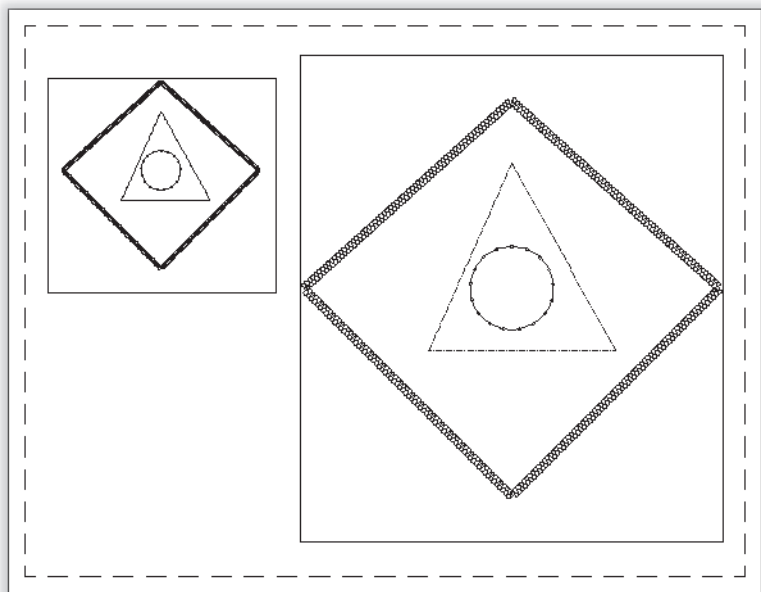
When set to **0**, the linetype scale in model space is independent of layout scale factors.



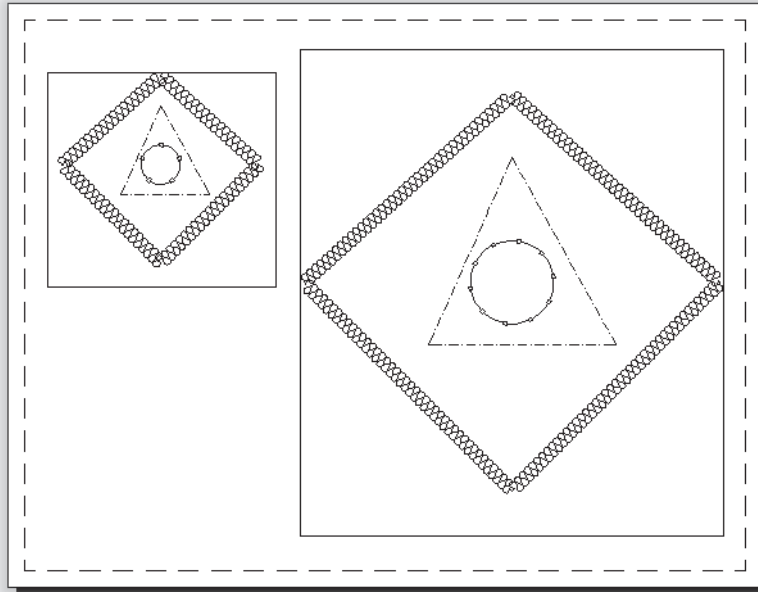
Note After changing the value of **PSLTSCALE**, you need to use the **REGENALL** command to update linetype scaling in all viewports.

TUTORIAL: MAKING LINETYPE SCALES UNIFORM

1. From the CD, open the *psltscale.dwg* file, a drawing of objects with several linetypes. In this drawing, **PSLTSCALE** is turned off (set to **0**). Notice that the linetypes in the two viewports have different scales, because the viewports have different scales.



2. To make linetype scales the same in all layout viewports, start the **PSLTSCALE** system variable:
Command: **psltscale** (Press ENTER.)
3. AutoCAD prompts you to change the value:
Enter new value for PSLTSCALE <0>: **1**
4. The change to **PSLTSCALE** has no effect until you regenerate all the viewports:
Command: **regenall** (Press ENTER.)
Notice that the linetypes have the same scale factor in both viewports.



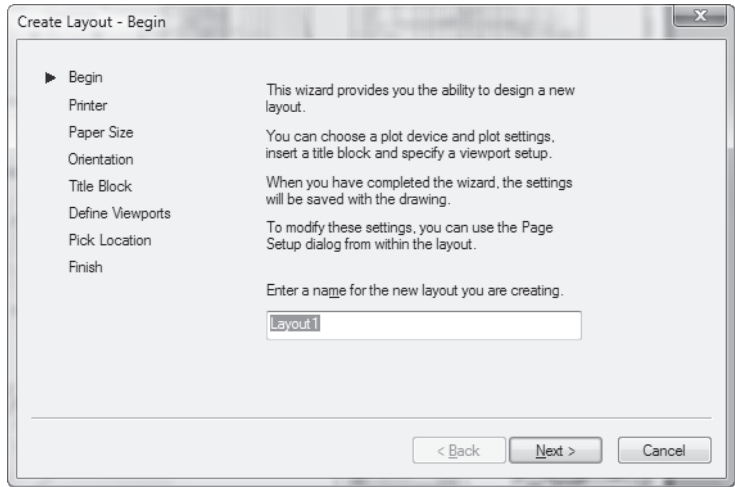
5. Change the linetype scale:
Command: **ltscale**
Enter new linetype scale factor <0.2500>: **.75**
Regenerating layout. Regenerating model.
The linetype scale changes in both viewports after AutoCAD automatically regenerates them. **VPMAX** shows the correct scale, whereas the **Model** tab does not.

LAYOUTWIZARD

The LAYOUTWIZARD command takes you through the steps of creating new layouts.

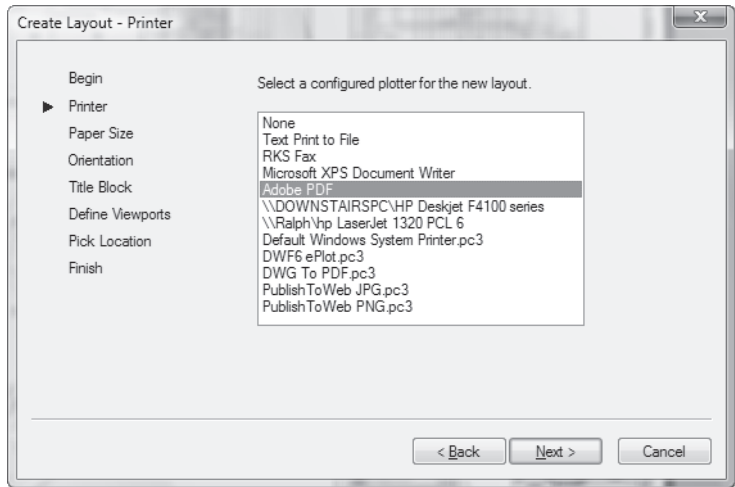
TUTORIAL: MANAGING LAYOUTS

1. To manage layouts, start the **LAYOUT** command:
 - From the Insert menu, choose **Layouts**, and then **Create Layout Wizard**.
 - From the Layouts toolbar, choose **New Layout**.
 - At the 'Command:' prompt, enter the **layoutwizard** command.
Command: **layoutwizard** (Press ENTER.)
2. In all cases, AutoCAD displays this first dialog box:



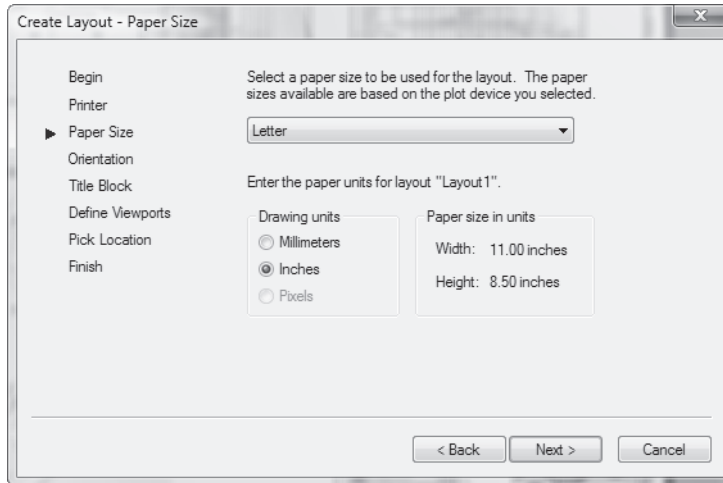
Enter a name for the layout, which will appear on the tab.
Click **Next**.

3. Select a printer from the list of Windows system printers and AutoCAD HDI printer drivers (shown as .pc3 files). You can select a different printer later, if need be.



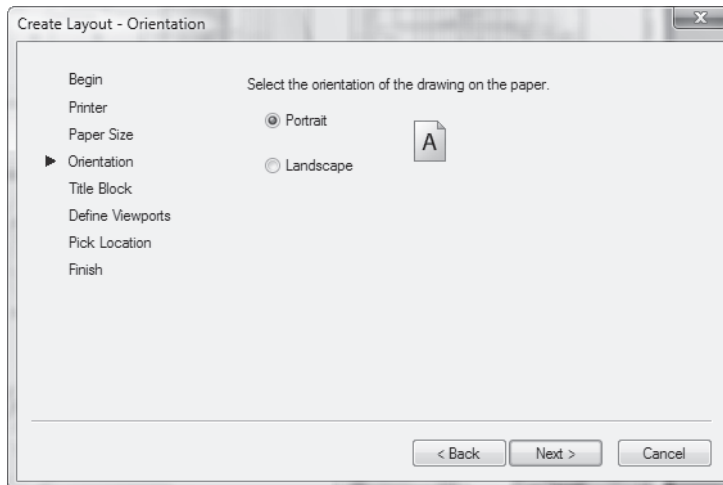
Click **Next**.

4. Select a paper size from the droplist. Only those sizes supported by the printer are listed.



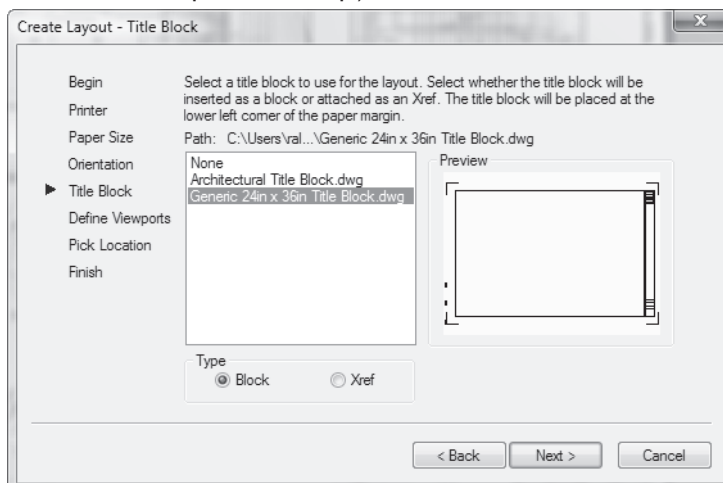
Drawing Units: Select the drawing units — inches or mm.
Click **Next**.

5. Select an orientation for the paper — landscape or portrait.



Click **Next**.

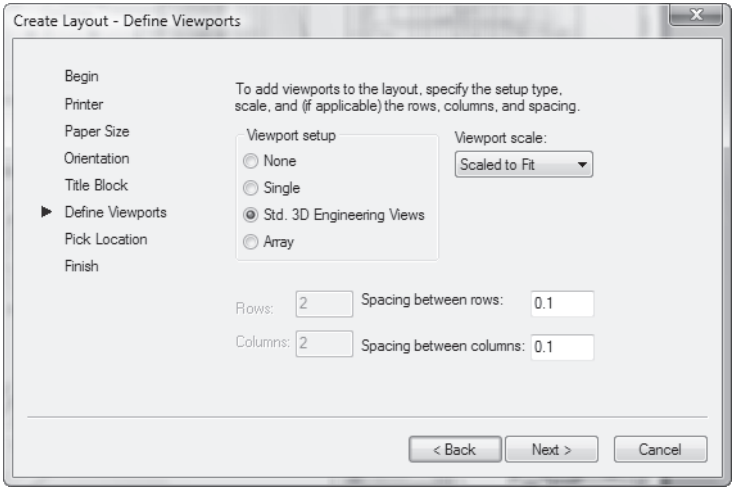
6. Select a drawing border/title block from the list.
(For A-size drawings, make sure the orientation of the border matches the orientation of the page you selected in the previous setup.)



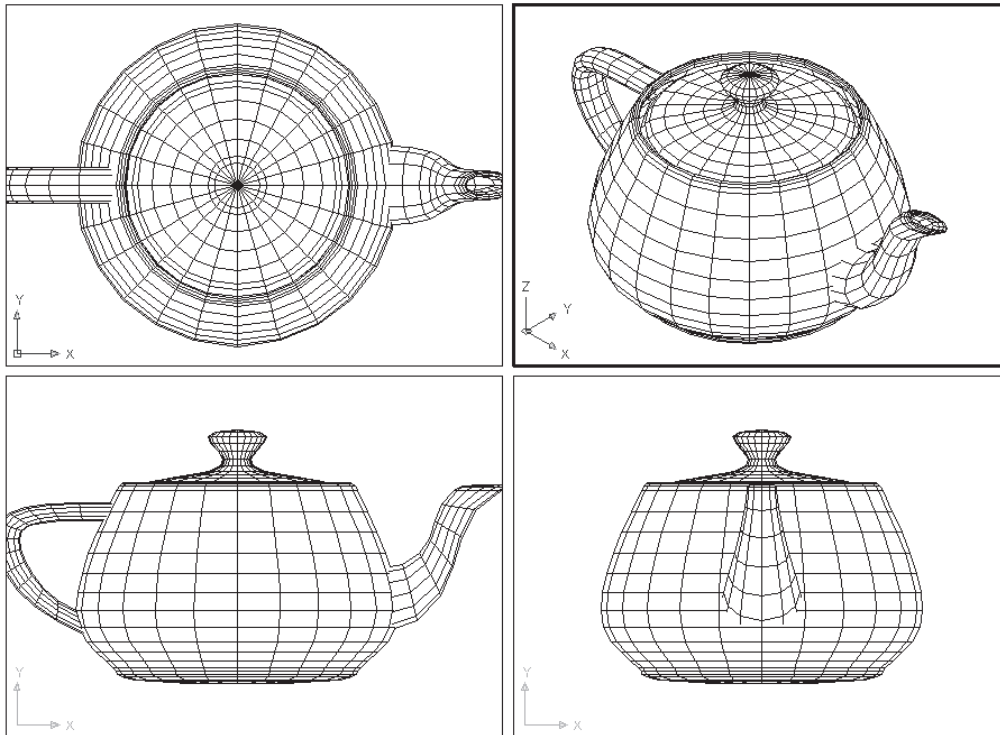
Type: Specifies whether you want the drawing border to be inserted as a block, or attached as an externally-referenced drawing (xref). If you are not sure, select **Block**. The following table provides you with the pros and cons of each:

Type	Pros	Cons
Block	Border and title block are part of the drawing, making it complete and easier to transport, such as by email.	Drawing size is larger. Blocks cannot be as readily updated as xrefs.
Xref	Drawing size is smaller. All xref'ed title blocks can be easily changed.	When sending the drawing to another office or client, you must ensure the xref is packaged with the drawing file.

7. **Viewport Setup:** Select the number and style of viewports. If you are not sure, select **Single**.



For engineering drawings, you would choose **Std. 3D Engineering Views**, which sets up the viewports and viewpoints illustrated below.

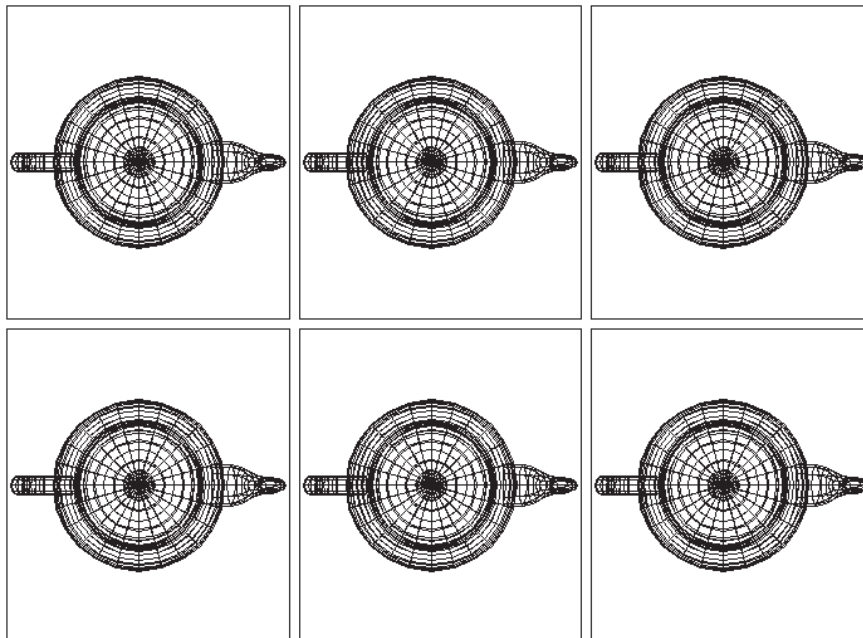


The four standard engineering viewpoints:

Top view — Isometric view

Side view — Front view.

The **Array** option sets up the viewports a rectangular array. You can specify the number of rows and columns, such as the 3x2 illustrated below.



A 3-column x 2-row array of viewpoints, each with a gap of 0.1 units from its neighbor.

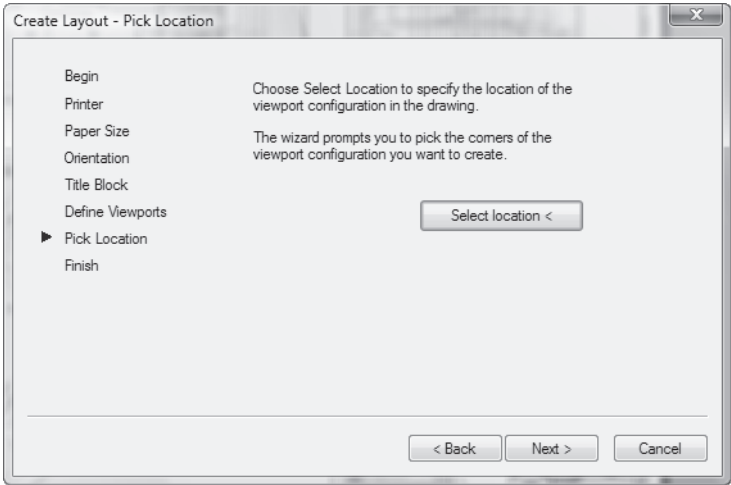
Viewport Setup	Comments
None	No viewport is created.
Single	One viewport is created.
Std. 3D Engineering Views	Four viewports are created, and then adjusted to show the standard engineering viewpoints of 3D drawings: front, side, top, and isometric.
Array	Rows x columns array of viewports is created.

Viewport Scale selects the viewport scale, which ranges from 100:1 to 1:100, and from 1/128"=1' to 1'=1'.

If you are not sure of the scale, select the **Scaled to Fit** option. (You can change it later by right-clicking the viewport border, selecting **Properties**, and then changing the value of **Custom Scale**.)

Optionally, you can also specify the gap between viewports; 0.1 units is the default.

- 8. Click the **Select Location** button to position the viewport(s).

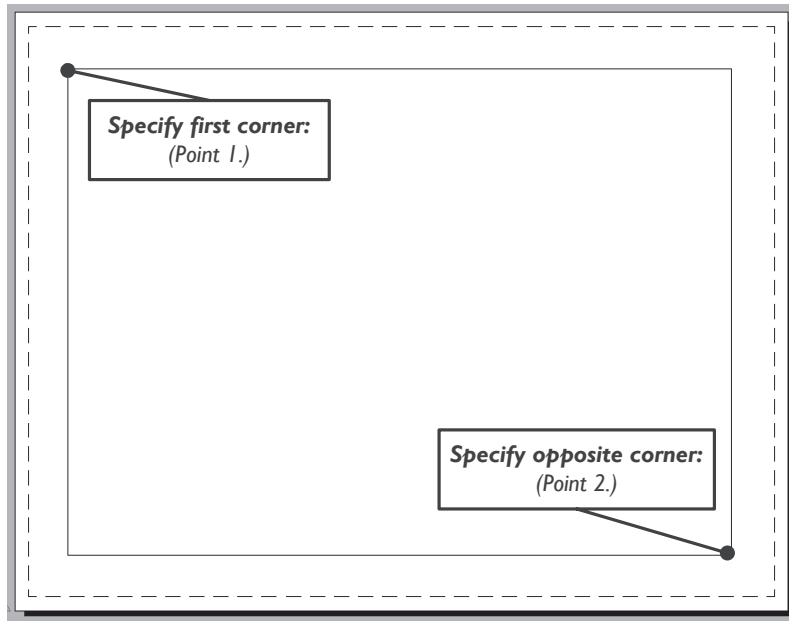


AutoCAD prompts you at the command line:

Specify first corner: *(Pick point 1.)*

Specify opposite corner: *(Pick point 2.)*

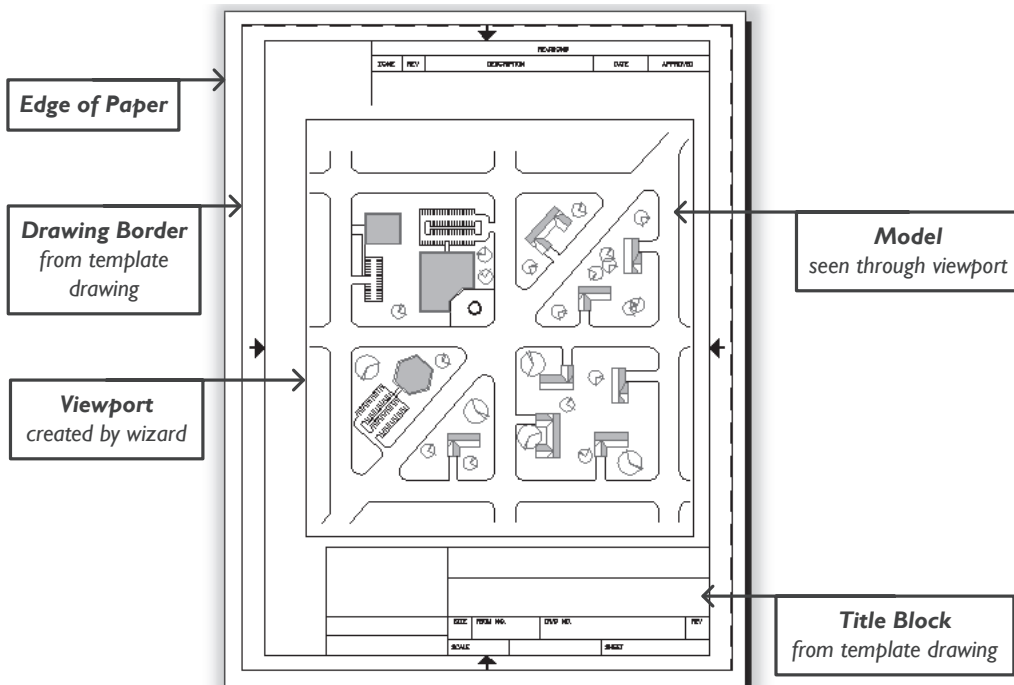
Pick two points to form a rectangle. AutoCAD later fits the viewport(s) to the rectangle.



Click **Next**.

9. Click **Finish**.

AutoCAD creates the viewport, which allows the model to show through into the layout, the elements of which are illustrated below. (The title block and number of viewports varies, depending on the options you selected during the wizard.)



 LAYOUT

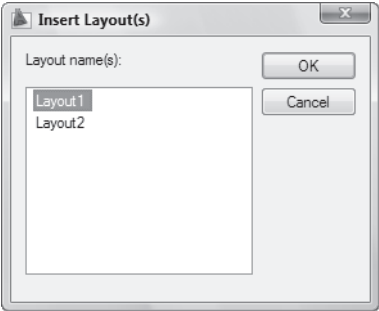
The LAYOUT command manages layouts.

TUTORIAL: MANAGING LAYOUTS

1. To manage viewports, start the **LAYOUT** command:
- At the ‘Command:’ prompt, enter the **layout** command.
Command: **layout** (Press ENTER.)

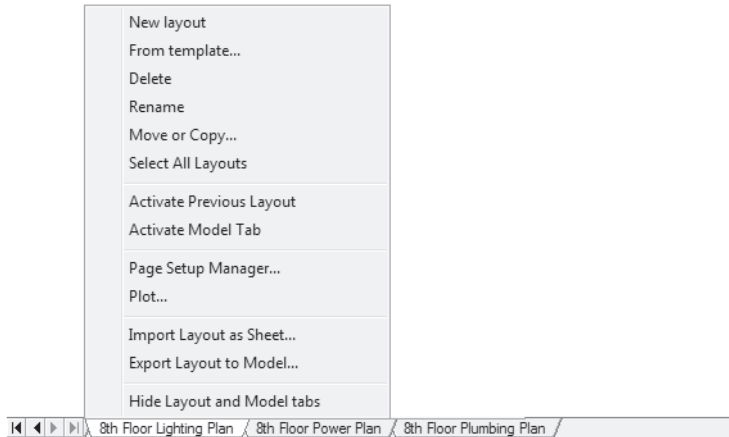
• As an alternative, enter the **lo** alias.
2. In all cases, AutoCAD displays the prompts at the command line:
Enter layout option [Copy/Delete/New/Template/Rename/SAveas/Set/?] <set>: (Enter option.)

Layout	Comment
Copy	Prompts you for the name of the layout to copy: Enter name of layout to copy:
Delete	Prompts you for the name of the layout to remove: Enter name of layout to delete: Note that the Model tab cannot be deleted. When you access this command through right-clicking the layout tab, the tab is immediately deleted.
New	Prompts you for a name, and then creates a new layout: Enter name of new layout: Names can be up to 255 characters long, but only a maximum of 31 are shown on the tab, fewer if there is less room. New tab are named “Layoutn.”
Rename	Prompts you for the name of the layout to rename: Enter name of layout to rename: Enter new layout name:
Saveas	Prompts you for the name of the layout to save as a .dwt template file: Enter layout to save to template:
Set	Prompts you for the name of the layout to make current. Enter layout to make current:
?	Lists the names of layouts.
Template	Creates new layouts based on .dwt, .dwtg, and .dxf files. Displays the Insert Layout(s) dialog box (illustrated below), and then inserts objects and layouts into the drawing.





Note Several more options are available through the shortcut menu, which are not found in the **LAYOUT** command. Right-click any layout tab, if they are visible:



Activate Previous Layout — returns to the previously-accessed layout, which is useful when a drawing has many layouts.

Activate Model Tab — returns to the Model tab.

Page Setup Manager — executes the **PAGESETUP** command.

Plot — executes the **PLOT** command.

Import Layout as Sheet — adds the current layout as a sheet; available only when the Sheet Set Manager palette is open.

Hide Layout and Model Tabs — “hides” the tabs by moving them to the status bar.



VIEWPORTS

The **VIEWPORTS** command creates and merges (removes) rectangular viewports.

When you switch to a layout for the first time, AutoCAD creates a single viewport automatically, which you can copy, move, and resize. To create additional viewports, use the **VIEWPORTS** command.

To create non-rectangular viewports, or convert objects into viewports, use **-VPORIS**, as described later. While both commands work in model space and in layouts, the non-rectangular viewports are available only in layouts.

These commands need two pieces of information: (1) the number of viewports, and (2) the location of the viewports.

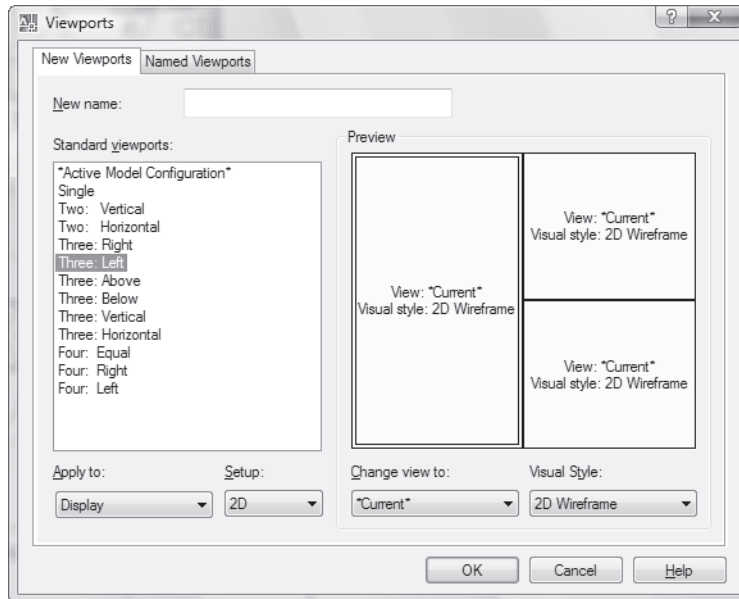
TUTORIAL: CREATING RECTANGULAR VIEWPORTS

- I. To create one or more rectangular viewports, start the **VIEWPORTS** command:
 - From the View menu, choose **Viewports**, and then **New Viewport**.
 - From the Viewports toolbar, choose the **Display Viewports Dialog** button.
 - At the 'Command:' prompt, enter the **viewports** command.

Command: **viewports** (Press ENTER.)

- Alternatively, enter the **vports** alias at the 'Command:' prompt.

In all cases, AutoCAD displays the Viewports dialog box.

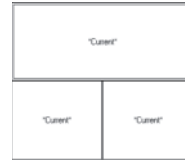


2. Select a style of viewport, as illustrated below.

Single



Three Above



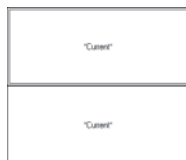
Two Vertical



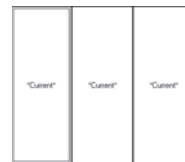
Three Below



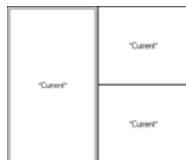
Two Horizontal



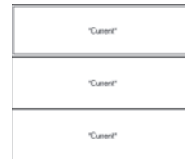
Three Vertical



Three Right



Three Horizontal



Three Left



Four Equal



3. Click **OK**. AutoCAD prompts you for the location of the viewports:
Specify first corner or [Fit] <Fit>: *(Press ENTER to fit the viewports within the display area.)*
4. Press **ENTER**, and AutoCAD fits the viewports to the display area.
Or, you can pick two points (forming a rectangle), and AutoCAD fits the viewports.



Notes Viewports created in layouts are called “floating,” because they can be moved. Each viewport is independent of the others. Polygonal (non-rectangular) viewports and viewports converted from objects can only be created in layouts.

Viewports created in model space are called “tiled,” because they stick together as tightly as tiles. They cannot be moved or copied. You can merge and erase them with the **-VPORIS** command.



-VPORIS AND VPCLIP



The **-VPORIS** command creates non-rectangular viewports, while the **VPCLIP** command clips viewports. You are not limited to viewports with rectangular or square shapes. AutoCAD can also create polygonal (non-rectangular) and circular viewports. These are sometimes called “clipped viewports,” because they usually hide a portion of the model.

-VPORIS converts closed objects into viewports, including closed polylines, circles, ellipses, splines, and regions. Polylines can be made of line and arc segments, and may intersect themselves.

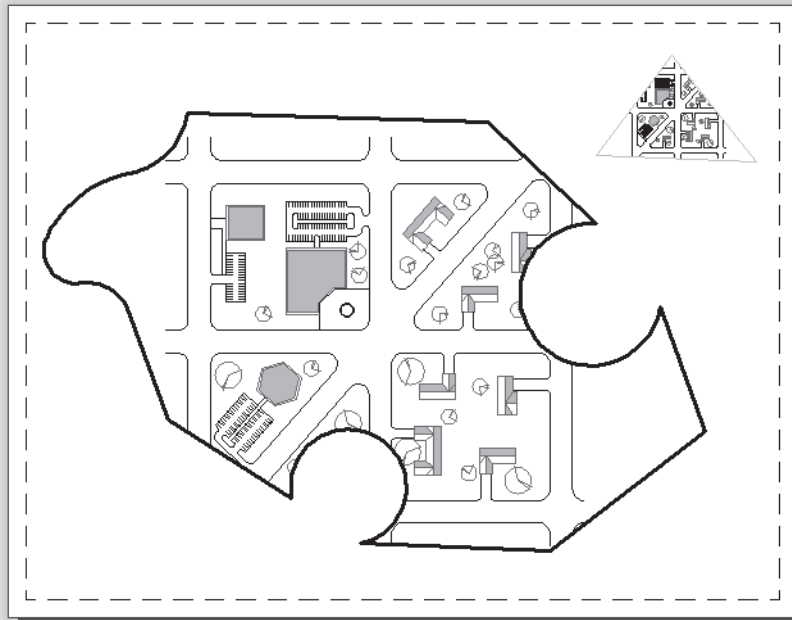
TUTORIAL: CREATING POLYGONAL VIEWPORTS

1. To create polygonal viewports, ensure AutoCAD is displaying a layout; this action cannot be carried out in model space.
2. Start the **-VPORIS** command:
 - At the ‘Command:’ prompt, enter the **-vpports** command.

Command: **-vpports** (Press ENTER.)

3. AutoCAD displays the prompts at the command line. Enter **P** to draw the outline of a polygonal viewport:

Specify start point or
[ON/OFF/Fit/Shadeplot/Lock/Object/**Polygonal**/Restore/Layer/2/3/4] <Fit>: **p**



4. Pick a point to start.

Specify start point: (Pick a point.)
5. The prompts that follow are exactly like those of the **PLINE** command. You can draw line segments and arcs. You must pick a minimum of three points, and then close the polyline.

An example of what is possible with polygonal viewport construction is illustrated above. There are two viewports: the larger one is made with line and arc segments; the smaller one in the upper right is made with the minimum of three line segments.



Notes In addition to clipping viewports, AutoCAD can also clip externally-referenced drawings and other attached objects with the **CLIP** command (new to AutoCAD 2010). If you prefer, you can use the following commands for specific file types:

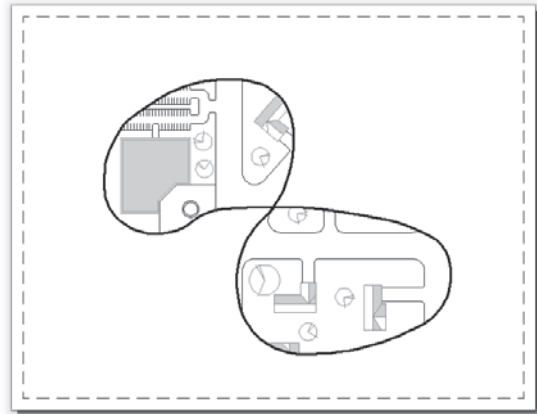
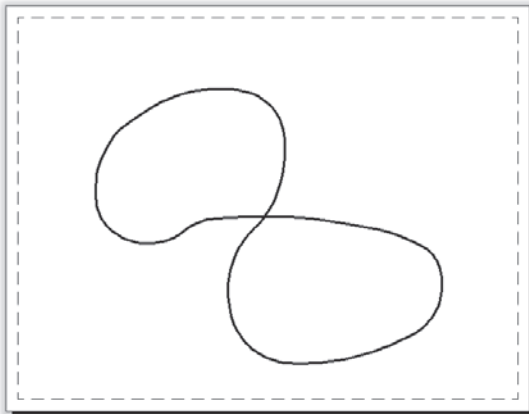
Xrefs — **XREFCLIP**
 Image files — **IMAGECLIP**
 DWF files — **DWFCLIP**
 DGN files — **DGNCLIP**
 PDF files — **PDFCLIP**

Viewports created as polygons and from objects cannot have their scales locked.

TUTORIAL: CREATING VIEWPORTS FROM OBJECTS

Unusually-shaped viewports can be made from objects. For this tutorial, ensure AutoCAD is displaying a layout; this action cannot be carried out in model space.

1. Draw a polyline, circle, ellipse, spline, or region. If drawing a polyline or spline, use the **Close** option to ensure the object is closed.
 The figure below illustrates a self-intersecting polyline drawn with the **PLINE** command, and then smoothed with the **PEDIT** command.

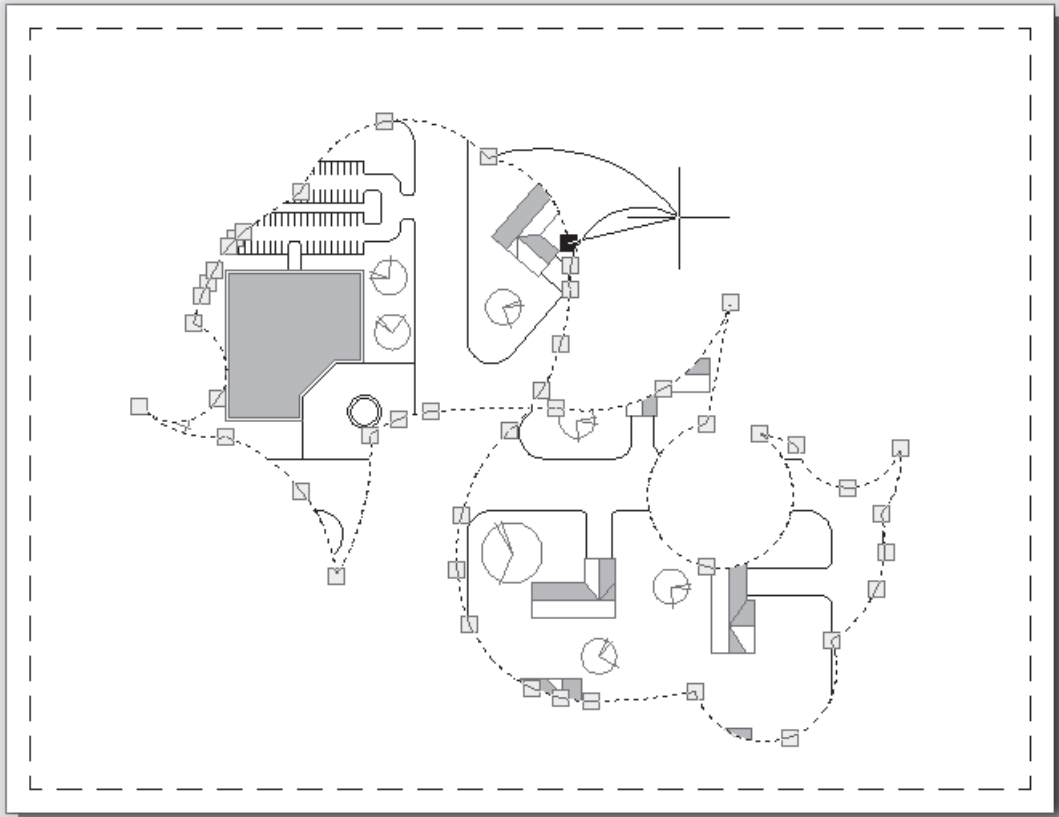


Left: Object drawn in paper space.
Right: Object converted to viewport boundary.

2. Start the **-VPOR** command:
 - At the 'Command:' prompt, enter the **-vpports** command.
 Command: **-vpports** (Press ENTER.)
3. Enter **O** to convert a closed object into a viewport:
 Specify corner of viewport or
 [ON/OFF/Fit/Shadeplot/Lock/Object/Polygonal/Restore/2/3/4] <Fit>: **p**
4. Pick the object to convert.
 Select object to clip viewport: (Select a closed object.)

Notice that AutoCAD converts the object into a viewport, and the model shows through, as illustrated above.

Once the viewport is in place, you can use grips editing to change the border, as illustrated by the figure below.



VPCLIP

The **VPCLIP** command operates in a manner similar to **-VPOR**: it clips rectangular viewports with polylines or objects. In addition, it can further clip a clipped viewport. Better than **-VPOR**, it converts clipped viewports back into rectangular ones.

- Command: **vpclip** (Press **ENTER**.)
- Select viewport to clip: *(Pick a viewport.)*
- Select clipping object or [Polygonal] <Polygonal>:

If you select a viewport previously clipped by this command, the **Delete** option also shows. The options are:

VpClip Option	Comment
Clipping object	Selects a closed object to be converted into a viewport boundary.
Polygonal	Picks points to designate line and arc segments for the viewport boundary.
Delete	Deletes the clipped viewport, and restores the rectangular viewport.

EXPORTLAYOUT

The **EXPORTLAYOUT** command copies layouts to model space.

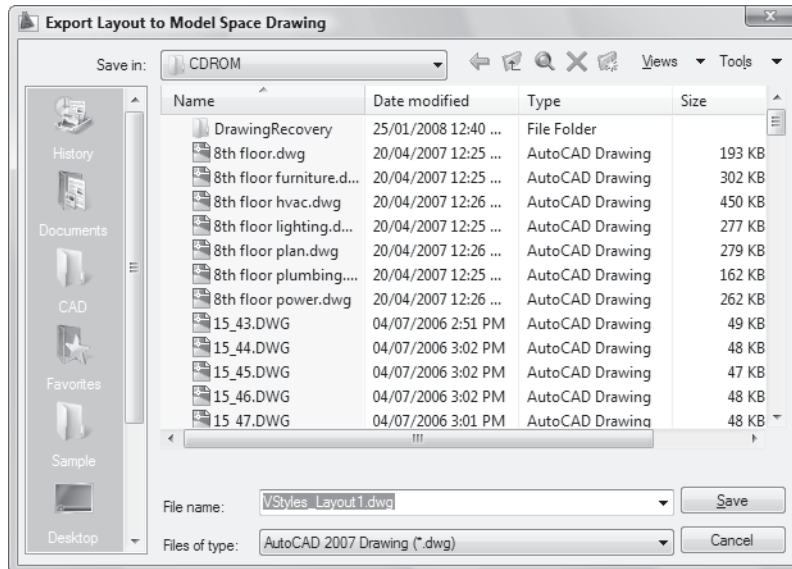
This command works by saving the layout as a *.dwg* file on disc, and then opening it as a new drawing in model tab. (It works only in layout mode, not when a viewport is maximized or in model tab.)

This command is useful for freezing layouts, since they are saved to separate drawings. Some CAD packages cannot read layouts, and so exporting them to model space makes them available to these packages.

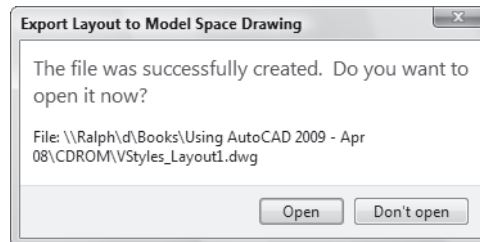
1. Open the *vstyles.dwg* drawing file from the Companion CD.
If necessary, switch to the **Layout1** tab.
2. Start the **EXPORTLAYOUT** command:
 - At the 'Command:' prompt, enter the **exportlayout** command.

Command: **exportlayout** (Press ENTER.)

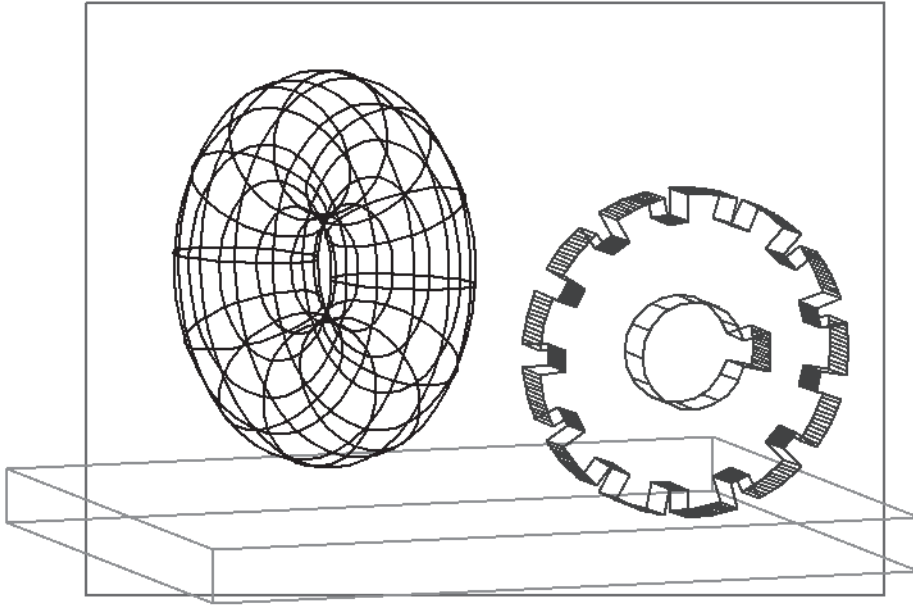
Notice the Export Layout to Model Space Drawing dialog box. The drawing has “_layout1” appended to its file name.



3. Click **Save**, and then wait while layout is converted to model space and saved on disk.



4. Click **Open**. AutoCAD opens the drawing in its own window.



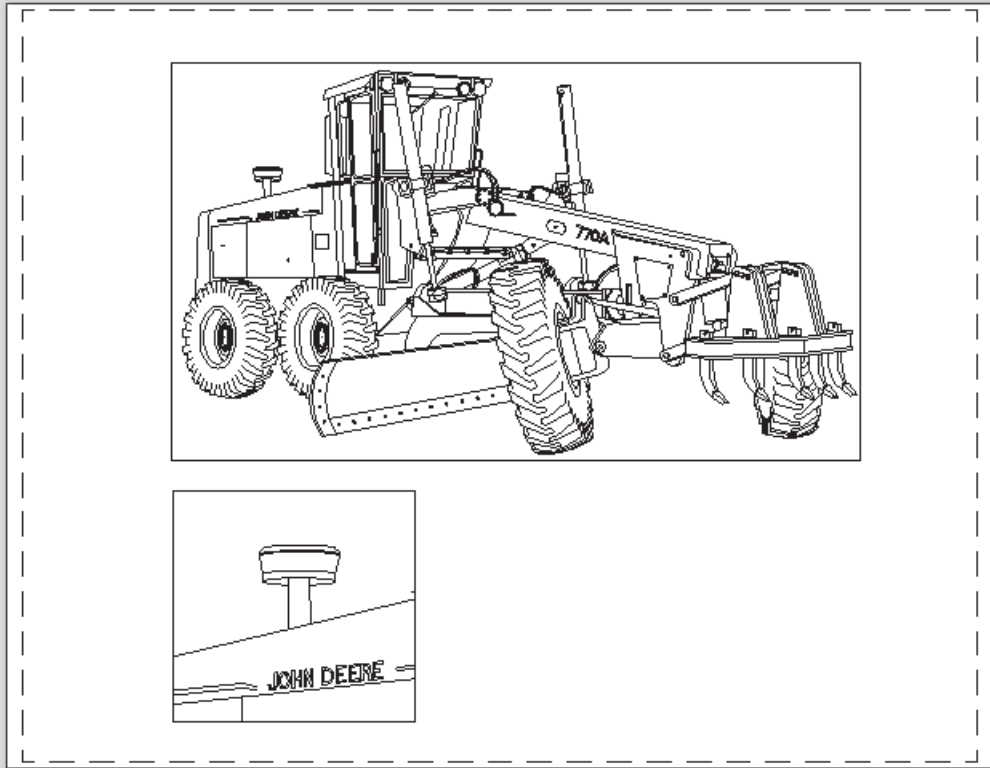
The viewport is mimicked by a rectangular polyline; 3D objects are still 3D. Notice that the drawing is scaled to be the same size as in layout mode, such as 10" wide.

(This means, the technical editor adds, that a circle of diameter 100 shown in a 1:2 viewport ends up having a diameter of 50 after exporting to model space. This could cause problems when sent to other CAD packages. He thinks Autodesk is allowing for viewports of different scales. The solution is to ensure the viewport is at 1:1 scale before running the `EXPORTLAYOUT` command.)

Portions of the drawing that extend beyond the viewport are not clipped. I have found that extremely complex layouts sometimes export incorrectly.

EXERCISES

- I. From the Companion CD, open the *grader.dwg* file, the 2D drawing of a grader. Click the **Layout I** tab, and create two viewports on an A0- or E-size sheet of paper:
 - a. The first viewport showing the entire grader, without the drawing border.
 - b. The second viewport showing a detail of the air intake.
 The result should look similar to the figure below. (All figures in these exercises are courtesy of Autodesk, Inc.)
 Save the drawing as *layout1.dwg*.



2. Start a new drawing, and then use the Layout Wizard to create a viewport with the following options:

Name	Practice Layout
Plotter	DWF 6
Paper Size	ANSI B
Drawing Units	Inches
Orientation	Landscape
Title Block	ANSI B
Viewport Setup	Single
Viewport Scale	Scaled to Fit

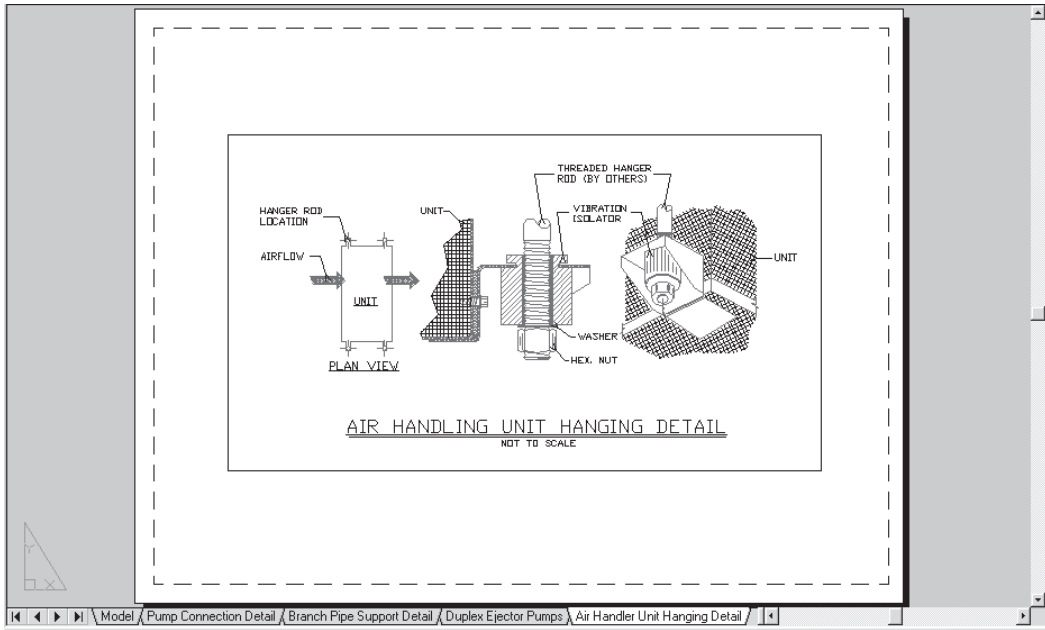
Save the result as *layout2.dwg*.

3. From the CD, open the *langer.dwg* file, a drawing detailing sump pumps. Create four layout tabs of the following names, which match the names of the details.
- Pump Connection Detail
 - Branch Pipe Support Detail
 - Duplex Ejector Pumps
 - Air Handler Unit Hanging Detail

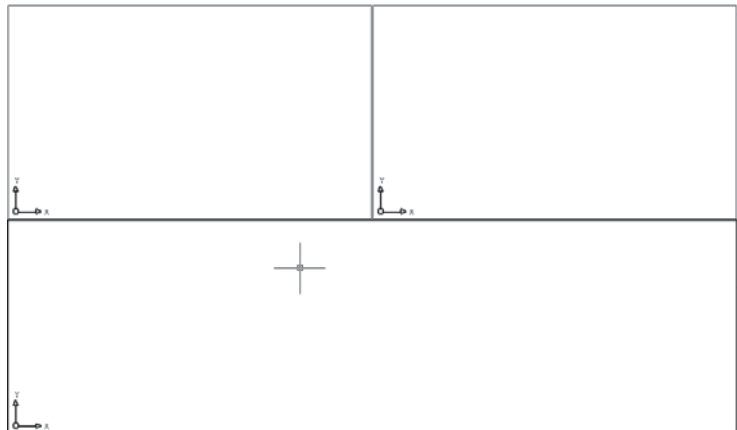
In each layout, place the namesake detail.

What is the viewport scale factor?

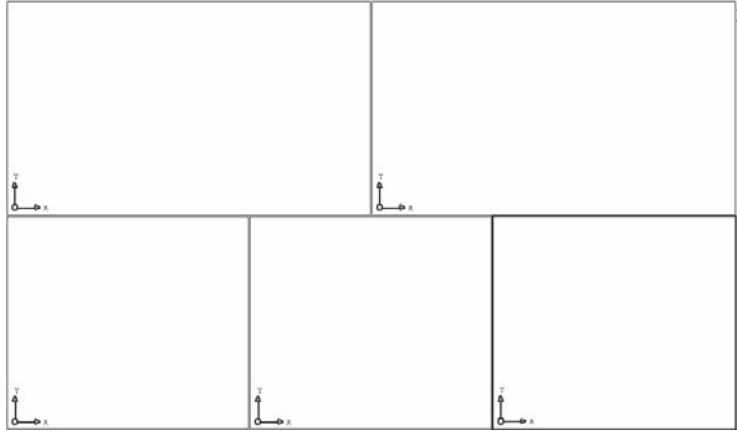
Save the drawing as *layout3.dwg*. The result should look similar to the figure below.



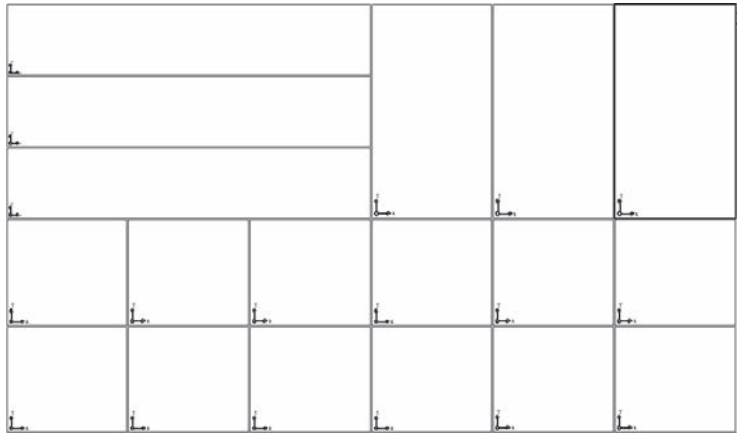
4. Use the **VIEWPORTS** command to create viewports with the following arrangements:
- a.



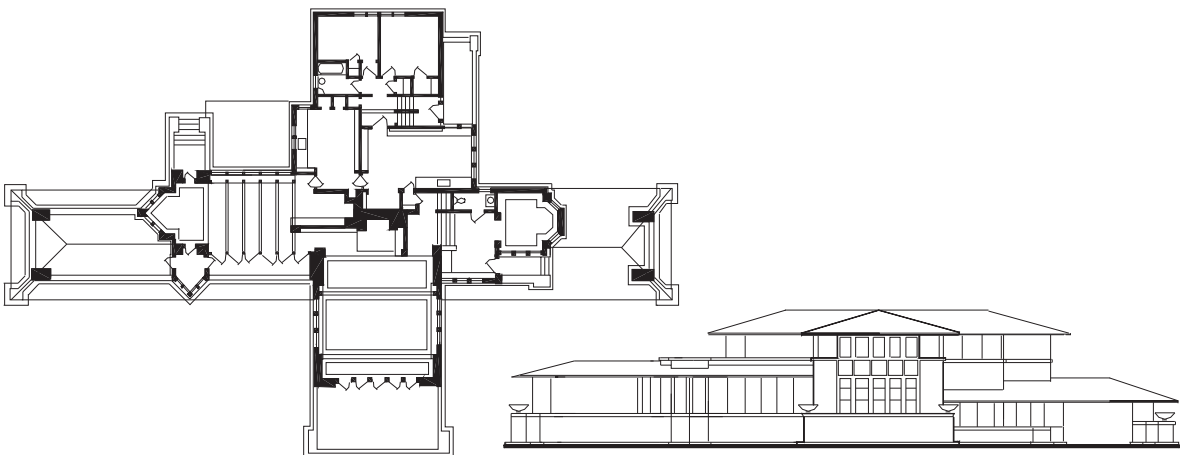
b.



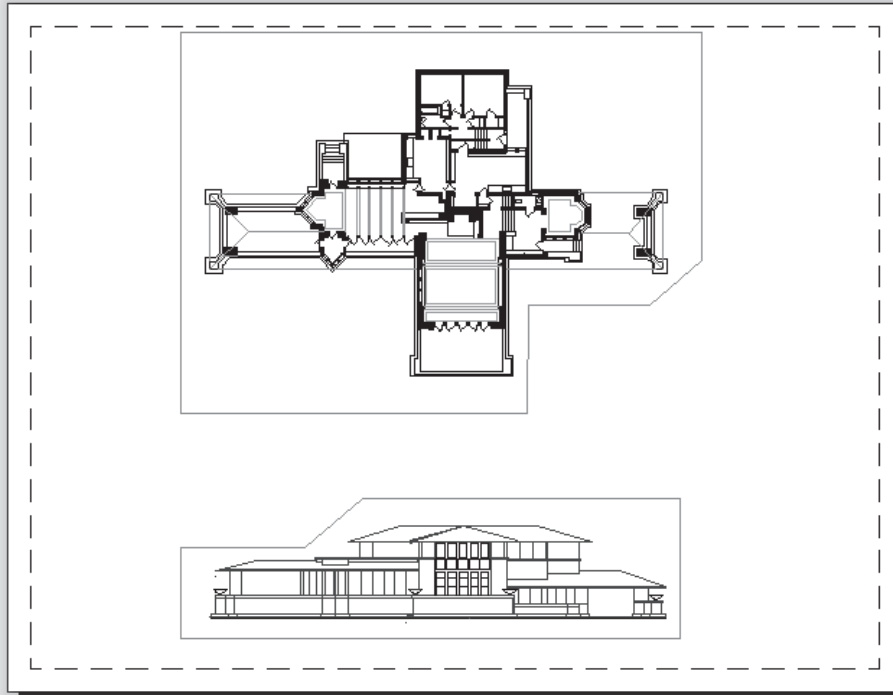
c.



5. From the Companion CD, open the *wright.dwg* file, a floor plan and elevation of a house designed by architect Frank Lloyd Wright.

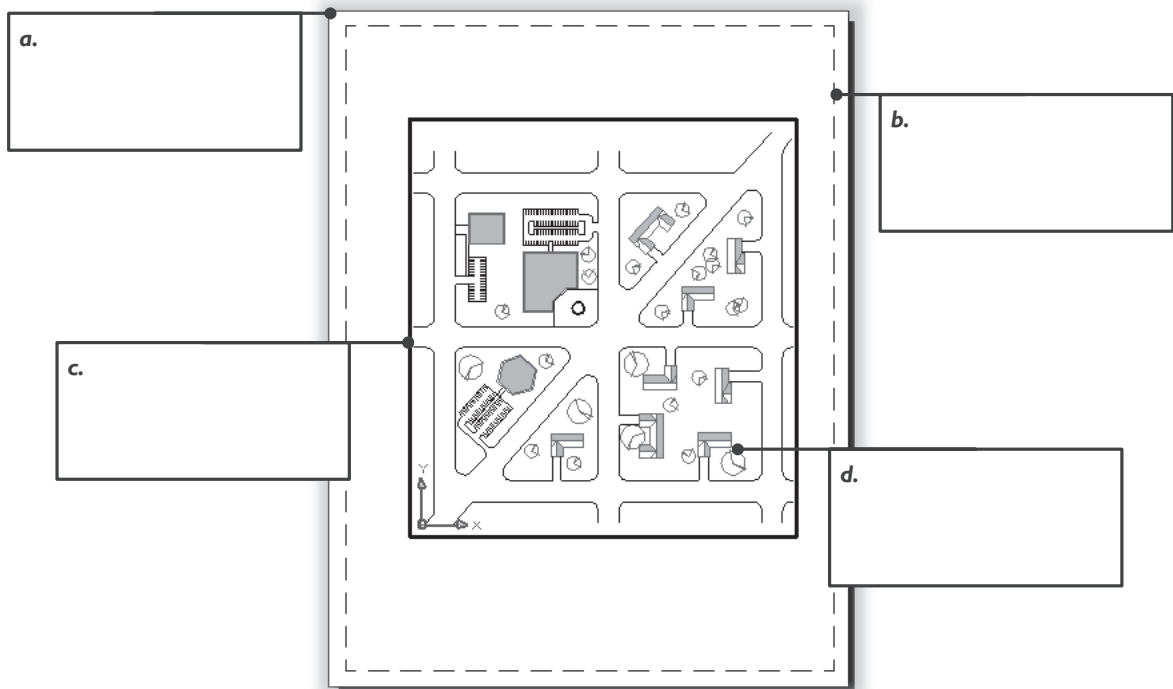


6. Create a pair of clipped viewports that show the two views independently, as illustrated below. Save the drawing as *layouts5.dwg*.



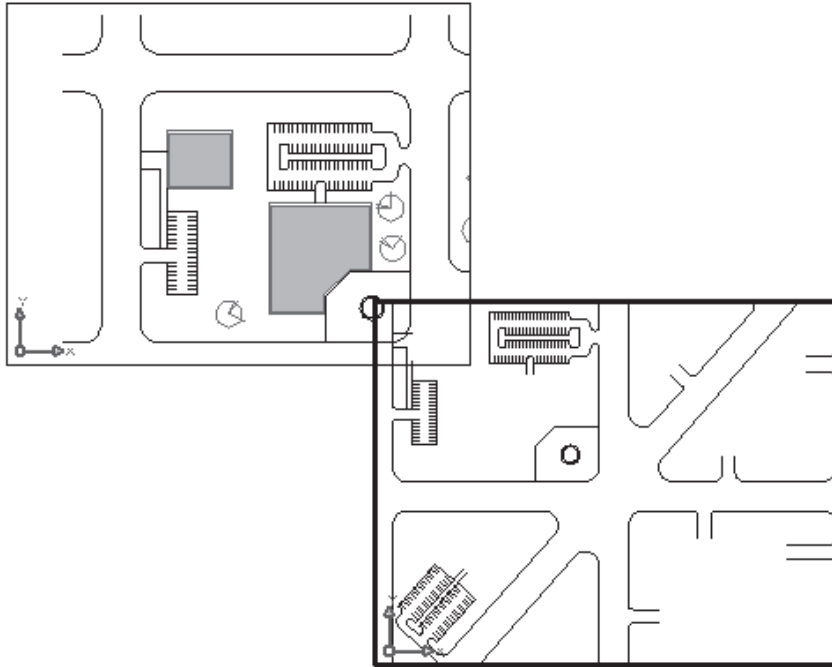
CHAPTER REVIEW

1. Explain the purpose of layouts.
2. Describe two ways to switch between model and layout mode:
 - a.
 - b.
3. In which tab are the following elements drawn full size?
 - The model
 - Title block
4. What does the layout represent?
5. Can a drawing have more than one layout?
6. How do you see the model in layouts?
7. Can viewports be modified?
 - Can viewports be copied?
8. Name the parts of the layout:
 - a.
 - b.
 - c.
 - d.



9. A layout has two viewports, of which one has a heavier border. What is the significance of the heavy border? What mode is AutoCAD in?
10. Explain the function of the following keystrokes when used with layout tabs:
 - CTRL+R**
 - CTRL+PGDN**

11. Describe the kind of drafting you can do when the layout is in:
PAPER
MODEL
12. Describe what happens when you resize a viewport in a layout?
When you copy a viewport.
When you select an object in one viewport.
13. Describe how to hide the viewport border.
14. What are the differences between the two viewports shown below? Explain a possible reason for the differences.



15. What are two ways to scale models in layouts?
 - a.
 - b.
16. What happens when you double-click a viewport?
17. Calculate the scale factor for the following viewport:
Drawing of automobile = 12', bumper to bumper length
Width of viewport = 10"
18. How would you move the model in the viewport?
19. Explain the difference between the **LAYER** command's **Current VP Freeze** and **New VP Freeze** options.
20. What does the snowflake mean in the Layer Properties Manager dialog box?
21. In which mode would you insert a title block?
PAPER mode of layout tabs.
MODEL mode of layout tabs.
Model tab.
22. What advantage does the **VIEWPORTS** command have over the **-VPORIS** command?
-VPORIS command over **VIEWPORTS** ?

23. Must viewports be rectangular?
If no, in what way?
If yes, why?
24. Describe the difference between:
Tiled viewports
Floating viewports
25. Briefly explain the function of the **PSLTSCALE** command.
26. When does **PSLTSCALE** take effect?
27. When is the **SPACETRANS** command commonly used?
28. What scale factor does **SPACETRANS** calculate?
29. Which command maximizes a viewport and also
Switches to model space?
Stays in paper space?
30. Describe one or more benefits to using the **VPMAX** command.
31. How can you add and remove scale factors from the VP Scale list?
32. When might you want to override layer properties in viewports?
33. List at least two ways of turning off layer property overrides.
34. What is the purpose of the **QVLAYOUT** command?
35. How might the **EXPORTLAYOUT** command be useful?

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CHAPTER 20

Plotting Drawings

The end product of CAD drafting is typically the drawing plotted on paper. In this chapter, you learn to plot drawings with these commands:

PREVIEW previews drawings before plotting.

PLOT plots drawings.

OPTIONS specifies default settings for plotting.

AUTOPUBLISH saves drawings in DWG and DWF or PDF simultaneously.

SCALELISTEDIT customizes lists of plot scales.

HIDEXREFSCALES hides plot scales imported from xrefs.

VIEWPLOTDETAILS reports on successful and failed plots.

PLOTSTAMP stamps plots with information about drawings and plotting.

PAGESETUP assigns plotters to layouts.

PUBLISH creates and plots drawing sets.

PLOTTERMANAGER creates and edits plotter configurations.

STYLESMANAGER creates and edits plot style tables.

NEW TO AUTOCAD 2010 IN THIS CHAPTER

- **PUBLISH** and **PLOT** exports PDF files with TrueType fonts, hatches, layers, and higher resolution.
- **PUBLISH** dialog box has been redesigned, now supports sheet lists and publishes all open drawings automatically.
- **AUTOPUBLISH** now supports PDF.

ALL ABOUT PLOTTERS AND PRINTERS

AutoCAD works with any printer connected to your computer and your network; it checks automatically for all system printers registered with Windows.

SYSTEM PRINTERS

A *system printer* is any local and network printer recognized by Windows. To see the list of system printers for your computer, choose the **Start** button on the Windows Taskbar, and then select **Printers**. Windows opens the Printers window, which lists the names of your computer's system printers.



The figure illustrates the Printers window for the author's computer. From left to right, the icons indicate:

Adobe PDF — the unadorned printer icon indicates a local printer. In this specific case, Adobe PDF creates PDF files from any document by “printing” it to a file.

HP LaserJet... — the tube represents “sharing,” and means the printer can be accessed by other computers on the network. To allow network access to one of your printers, right-click its icon, and then select **Sharing**; in the Properties | Sharing dialog box, select the **Shared As** radio button, and then choose **OK**.

HP Photosmart... — the check mark on this icon indicates the default printer. AutoCAD and other programs use this printer automatically, unless you specify a different one. To select a different printer as the default, right-click one of the other printer icons, and then select **Default Printer** from the shortcut menu.

To change the properties of a system printer, right-click its icon, and then select **Properties**. The content of the Properties dialog box varies, depending on the printer's capabilities. Commonly, though, you can set the default resolution, paper size and source, color management, and so on.

Nontraditional Printers

AutoCAD also works with nontraditional printers, the most common today being Adobe Acrobat PDF (short for “portable document format”). AutoCAD creates PDF files of drawings with the **EXPORTPDF** command.

Autodesk promotes the DWF format for sharing drawings over the Internet. The **3DDWF**, **PUBLISH**, and **EXPORTPDF** commands create *.dwf* files from drawings. In addition, Autodesk also provides the free DWF Writer software that creates *.dwf* files from any application; you can download it from www.autodesk.com/dwfwriter.

Less popular today are faxes. If computers have modems, then fax capability is included. Windows lets you fax from any software program, including AutoCAD, provided the computer is connected to the telephone system. (Not all versions of Windows include the fax software, so you may have to purchase it separately.) The process is as simple as selecting the fax as the printer. The drawback to faxing is that AutoCAD typically splits large drawings onto multiple A- or A4-size sheets of paper (roughly 8" x 11" each).

LOCAL AND NETWORK PRINTER CONNECTIONS

Printers usually connect to computers through USB (universal serial bus) connections. Computers can theoretically have up to 128 USB ports, but two to eight are typical. To support older printers, some computers still have *parallel* ports; the very oldest printers and pen plotters use *serial* ports, which operate slowly and are difficult to configure.

Network printers are connected directly or indirectly to the network. If connected *directly*, the printer contains its own network card; if *indirectly*, the printer is connected to a computer, and then through the computer's network card to the network.

Computers connected to networks can print to any network printer — provided the computers have been given permission to access the network printers. This works in reverse, too: you can restrict other networked computers from using your computer's printer.

Differences Between Local and Network Printers

The primary difference between local and network printers is how Windows sees them. During printer setup, you tell Windows whether the printer is Local or Network, so that it knows whether to search for it on your computer's local ports, or along the network.

Another difference is that local printers are typically more available. A network printer might be inaccessible because the network is down, or because too many other computers are sending files.

The advantage to networking printers is that everyone in the office can share all printers. If the printer attached to your computer breaks down (through mechanical failure, lack of paper or ink, and so on), you can easily access another one.

ABOUT DEVICE DRIVERS

A term you will occasionally read is “device driver.” Without device drivers, software programs cannot communicate with printers, graphics boards, and other computer hardware — collectively known as “peripherals.”

To understand the role of device drivers, it helps to know their history. In the early days of personal computers, there were very few choices for external peripherals, and the peripherals had very few abilities. For instance, early graphics boards and printers only output monochrome text — no color, no graphics!

As personal computers became popular in the late 1980s, more and more companies began inventing add-ons for computers. Graphics boards and printers gained the ability to output graphics, and then added color. Varieties of input devices flourished, such as digitizing tablets and 100-button keypads.

Peripherals vendors typically took one of two approaches to make their products work with operating systems. One approach was to mimic a competitor's protocol, such as HPGL (Hewlett-Packard Graphics Language) for plotters and TIGA (Texas Instruments Graphics Architecture) for graphics boards. If the standard lacked the desired capabilities, then the second approach was to create a new standard.

Even with standards in place, peripherals still needed a way to communicate with the operating system. Initially, each peripheral vendor wrote its own software, called a “device driver,” software that drives the devices — or, “drivers” for short.

When Microsoft began marketing Windows, they decided to include as many drivers as possible to reduce the resistance to switching from DOS to Windows. Microsoft undertook the thankless job of writing thousands of drivers for nearly any peripheral on the market — albeit in many cases, one device driver served several related hardware products.

Initially, Autodesk undertook the same approach. The company wrote AutoCAD drivers for all peripherals, and even boasted about the long list. By the mid-1980s, however, the job became overwhelming, and so Autodesk began to require that peripheral vendors write the drivers themselves

using the ADI kit supplied by Autodesk — the AutoCAD Device Interface. In the 1990s, the capabilities were expanded by the enhanced HDI kit — Heidi Device Interface.

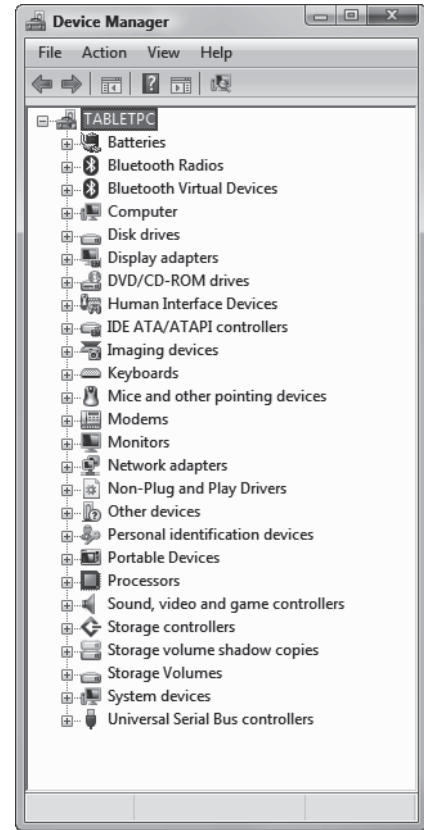
(HEIDI is short for “HOOPS Extended Immediate-mode Drawing Interface.” HOOPS, in turn, is short for “Hierarchical Object-Oriented Picture System.” So really, HDI is short for Hierarchical Object-Oriented Picture System Extended Immediate-mode Drawing Interface Device Interface — whew!)

Drivers Today

Today, Autodesk provides drivers for a number of plotters and two graphics board standards, OpenGL and DirectX. It relies on Microsoft to provide drivers for common peripherals, such as mice, and on hardware vendors for specialized peripherals.

From time to time, hardware companies update device drivers to fix bugs and enhance capabilities. They typically provide free downloads of the latest drivers. If plots don’t work correctly, the first thing to suspect is a device driver that needs updating. On the other hand, if the driver is working well, *don’t upgrade*, because sometimes upgrades can cause new problems.

To see the list of drivers installed on your computer, open the Control Panel, and then double-click the **System** icon. (In some versions of Windows, select the **Hardware** tab.) And then click **Device Manager** to see a list similar to that illustrated at right.

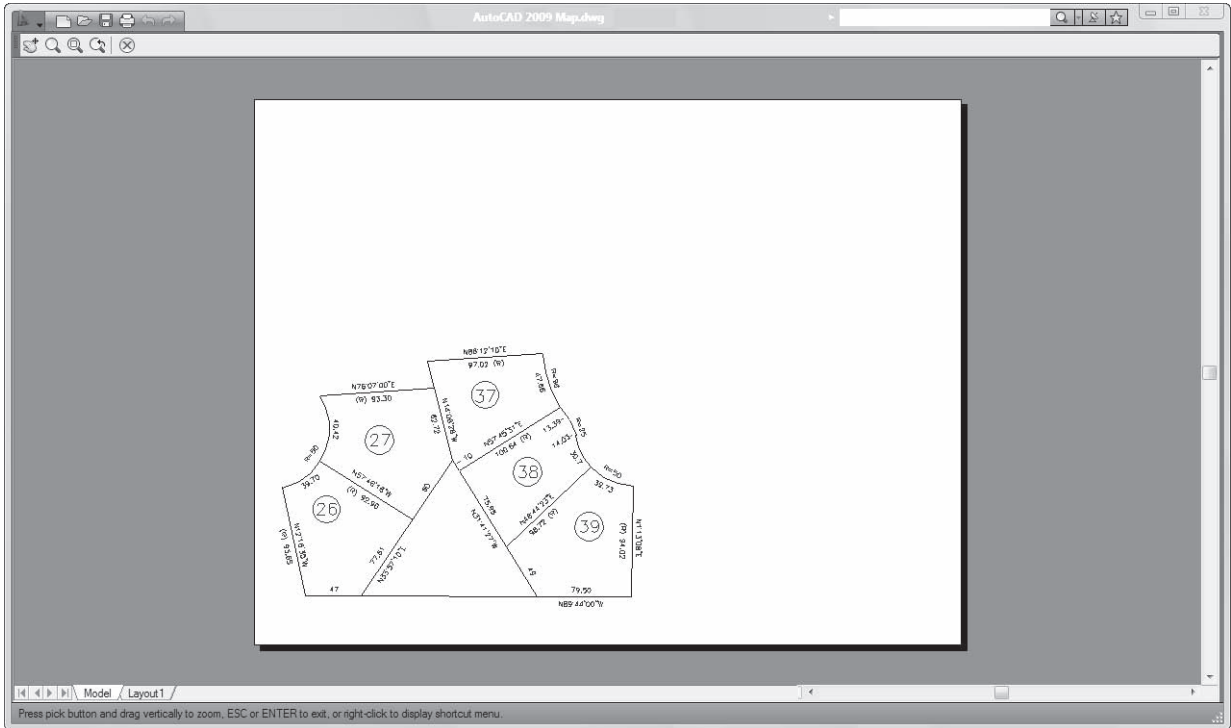


With this background information in place, let’s see how AutoCAD handles the plotting of drawings. In general, the process is as follows:

1. Start the **PLOT** command and set the plotting options.
2. Use the **PREVIEW** command to check that the plot will turn out correctly.
3. Send the drawing to the printer or, if necessary, go back to the Plot dialog box to correct settings.

The rest of this chapter concerns itself with setting up plotters by a variety of methods.

2. Once the preview is generated, AutoCAD displays it in the preview window. (If the drawing is complex, a dialog box may appear showing AutoCAD's progress in generating the preview image.)



3. Press **ESC** to exit preview mode and return to the drawing.

PREVIEWING PLOTS: ADDITIONAL METHODS

The preview shows the drawing on a white rectangle that represents the paper. (The color of the area that represents the sheet of paper can be changed: use the **OPTIONS** command's **Display | Colors** button, and then select **Plot Preview** from the Context list.)

Controlling the Preview Image

Once in the preview window, you can zoom and pan the image.

 The cursor initially looks like a magnifying glass. As you drag the cursor up and down (by holding down the left mouse button), the preview zooms in and out.

For other viewing options, you can use the toolbar, or else right-click the drawing for a shortcut menu of viewing choices:



 When you select **Pan** from the toolbar or shortcut menu, the cursor changes to a hand . Hold down the left mouse button and the view moves (pans) as you move the mouse around.

 The **Zoom Window** option operates identically to the regular **ZOOM** command's **Window** option.

The cursor changes to ; pick two points to identify the area to enlarge.

 The **Zoom Original** option returns the original view, much like the **ZOOM Extents** command.

 To plot, click the **Plot** button; otherwise, click  **Cancel** to return to AutoCAD. Alternatively, press ENTER, the spacebar, or ESC to exit the preview window.

(Although the preview screen has Model and Layout tabs, they are inoperable.)

In summary, Preview's toolbar provides the following options:

Icon	Shortcut Menu	Cursor	Meaning
	Exit	...	Exits preview.
	Plot	...	Plots the drawing.
	Pan		Switches from zoom to pan mode.
	Zoom		Switches from pan to zoom mode.
	Zoom Window		Zooms into a windowed area.
	Zoom Original	...	Returns to normal view.

PLOT

The **PLOT** command displays the Plot dialog box, and then plots the drawing.

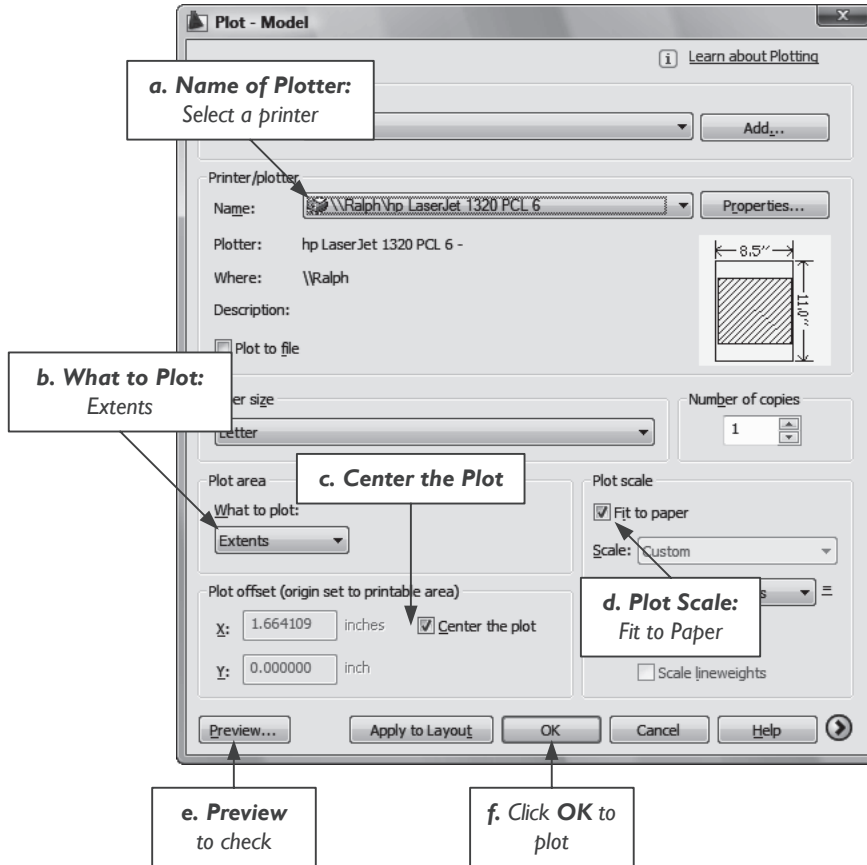
The dialog box lists all of the options AutoCAD needs resolved before outputting drawings on paper or to files. Drawings can be plotted to printers, plotters, or to files.

TUTORIAL: PLOTTING DRAWINGS

- I. To plot drawings, start the **PLOT** command:
 - On the keyboard, press the **CTRL+P** shortcut.
 - Right-click a layout tab, and select **Plot** from the shortcut menu.
 - In the ribbon's Output tab, click the **Plot** button in the Plot panel.
 - At the 'Command:' prompt, enter the **plot** command.
 Command: **plot** (Press ENTER.)
 - Alternatively, enter the aliases **print** or **dwfout** (older commands that have been integrated into plotting) at the 'Command:' prompt.

In all cases, AutoCAD displays the Plot dialog box.

2. The Plot dialog box has many options, and it can be confusing to navigate. For your first plots, I recommend you specify the following information as a minimum:



- Printer/plotter:** Select the name of a printer or plotter. In many cases, this is whatever printer is hooked up to your computer (the “system” or “default” printer).
 - What to Plot:** select **Extents** to ensure that every object is included in the plot.
 - Plot Offset:** select **Center the plot**. (If the plot is not centered, it may be because **What to Plot** is set to **Display**.)
 - Plot Scale:** select **Fit to Paper** so that nothing is printed off the edge of the page.
 - Click the **Preview** button to check that the plot will turn out.
 - Click the **OK** button to start the plot. AutoCAD plots the drawing.
3. If you wish only to set up the plotting parameters, but not actually plot, then click **Apply to Layout** to save the changes, and then **Cancel** to return to the drawing editor.



Notes AutoCAD does not plot layers that are frozen or have the “No Plot” property. As well, AutoCAD does not plot the Defpoints layer. If part of your drawing does not plot, it could be because you drew on the Defpoints layer.

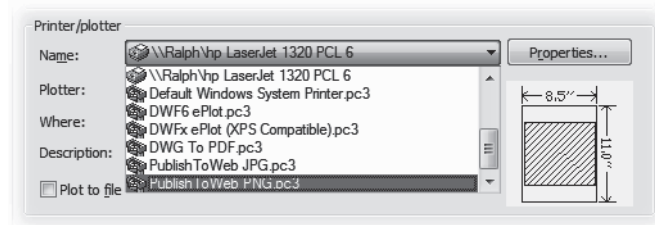
To create 2D DWF files of drawings, use the **EXPORTDWG** command; to create multi-sheet 2D DWF files, use **PUBLISH** and its **DWF File** option; to create 3D versions of DWF files, use the **3DDWF** command.

Here in greater detail are the steps to plot successfully.

Select a Plotter/Printer

Your computer may have two or more printers available for plotting. The Printer/Plotter droplist contains the list of printers available to you — local and networked, as well as special drivers for plotting to files, such as PDF and DWF.

From the **Name** droplist, select a printer name.



If the printer you want is not listed here, read how to add it with the **PLOTTERMANAGER** command, described later.

To change properties specific to the printer, such as its resolution or color management, choose the **Properties** button.

Plot to File

Some “plotters” listed by the Plot dialog box are designed only to plot to file. These include “DWF6,” “DWG to PDF,” and “Publish to Web.” For them, the **Plot to File** option is turned on automatically and grayed-out, because it cannot be turned off.

To make PDF files of drawing, choose the “DWG To PDF” plotter. These can be viewed by the free Acrobat Reader software from Adobe. Added to AutoCAD 2009 was “DWFx ePlot (XPS Compatible),” which creates DWFx files that can be read by Internet Explorer v7/8 running on Vista or 7.

The **Plot to File** option saves drawings to files. *Plot files* can be imported into other software. For example, many brands of plotters and graphics programs can import files created by HPGL plotter drivers, usually meant for Hewlett-Packard brand plotters.

(The technical editor provides this tip: Plot-to-file is handy when you are on the road and need access to someone else’s printer. Connecting to foreign printers directly can be a hassle, because it involves cables and installing printer drivers, which might not be available for your computer and its operating system. Instead, plot to a USB stick, and then copy the file to the printer.)

Here’s how to create plot files:

1. Click **Plot to File** so that a check mark appears.



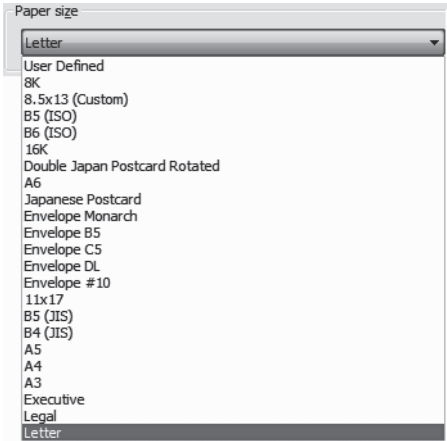
2. Click **Plot**.
3. Instead of plotting the drawing, AutoCAD displays the Browse for Plot File dialog box. Select a folder, and if necessary change the file name.
4. Click **Save**.

AutoCAD generates the plot file, saving it under the name of the drawing and using the file extension of *.plt*.

Select the Paper Size

Paper Size specifies the dimensions of the paper (media) used by printers. AutoCAD knows the appropriate media sizes for the selected printer. Thus the list of sizes you see here varies according to the printer. Make your selection from the droplist, and ensure the same size of paper is loaded into the printer.

Most desktop printers use Letter paper in North America; drafters call it “A size.” It measures 8.5x11". In Europe and other parts of the world that use metric sizes, A4 paper is the equivalent; it measures 8.26x11.7".



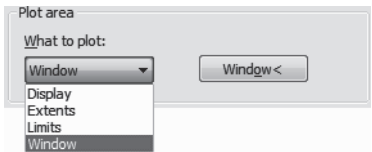
If the paper size you need is not listed, you can add it with the PLOTTERMANAGER’s **Custom Paper Sizes** option.

Sometimes printers will not print drawings. This may occur when drawings are set up with a printer or size of paper that is unavailable.

Select the Plot Area

Many times you want to plot the entire drawing — known as the “extents.” In some cases, however, you may want to plot just specific parts of drawings.

AutoCAD offers the Plot Area droplist with several options for determining how much of the drawing to plot:

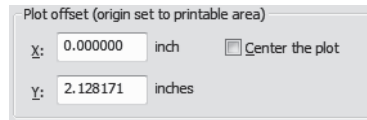


Plot Area	Comments
Display	Plots the current view of drawings — the “what you see is what you get” plot.
Extents	Plots the extents, a rectangle that encompasses every part of the drawing containing objects; like performing a ZOOM Extents, and then plotting with the Display option.
Limits	Plots the limits of model space, as set by the LIMITS command.
Window	Plots a rectangular area identified by two windowed picks. After clicking Window, you are prompted to pick the two corners of a rectangle: Specify first corner: (Pick a point or type X,Y coordinates.) Specify opposite corner: (Pick another point or type X,Y coordinates.)
Layout	Plots the extents of the page in layouts (available only when plotting layouts).
View	Plots named views; available only when the drawing contains named views (created with the VIEW command). Select View, and then select a view name.

Plot Offset

The *plot origin* is at the lower left corner of the media for most plotters, just as it is for AutoCAD drawings. Some plotters, however, locate their origin at the center of the media; AutoCAD normally adjusts for them.

Sometimes you need to shift the plot on the media, for example to avoid a preprinted title block. Use the **Plot Offset** section of the Plot dialog box:



- Positive values shift the plot to the right and up.
- Negative values shift the plot to the left and down.

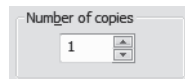
In most cases, however, you will probably choose the **Center the plot** option to center the plot on the media.



Note Changing the plot's offset may result in a *clipped* plot, where part of the drawing is not plotted, because it extends beyond the edge of the media's margin.

Number of Copies

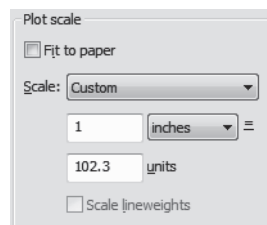
Most times, you want just one copy of the plot. If you need more, however, increase the **Number of Copies** counter.



The largest number you can dial in is 99. When plotting to file, number of copies is limited to one.

Plot Scale

To plot the drawing at a specific scale, select one of the predefined scale factors provided in the Plot Scale list box. These range from 1:1 to 1:100 and 1/128"=1'.



For draft plots, select **Fit to Paper**. Together with a plot area of **Extents**, this ensures your entire drawing fits whatever size of media you select.

Another choice is **Custom**, where you specify the scale factor. Specify the number of *inches* (or millimeters) on the paper to match the number of drawing *units* to be plotted. For example, a scale of 1" = 8'-0" means that 1" of paper contains 8'-0" (or 96") of drawing.

To set the custom scale correctly, enter:

- Inches **1"**
- Units **96"**

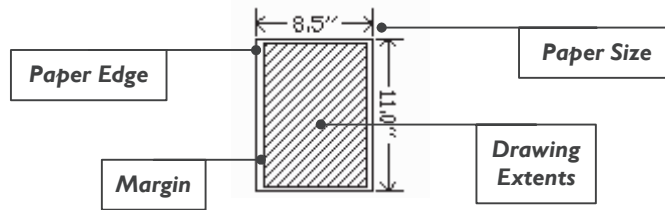
This is the same as a scale of $1/8" = 1'-0"$. The example assumes that the drawing units are set to architectural units. You can customize the list of plot scales with the `SCALELISTEDIT` command. (This command also affects annotative scale factors and the viewport scale list in the Properties palette.)

Preview the Plot

You should preview the plot to see the area of the paper on which the plot appears. AutoCAD's plot preview allows two types of preview: partial and full.

Partial Preview

The *partial* preview does not show the drawing elements but rather the position of the plot on the paper. It's called "partial," because it shows only the basics. (*History:* this option was more desirable in decades past, because slow computers took a long time to generate full previews.)



The paper's edge is shown by the outer rectangle. To see the paper size and printable area (a.k.a margin), pause the cursor over the partial preview image.

The margin is the non-printing area outside of the drawing; its size varies according to the capabilities of the printer and the settings in the Plot dialog box: Plot area, Plot offset, and Plot scale.

The area covered by the drawing is shown by the diagonal hatching.



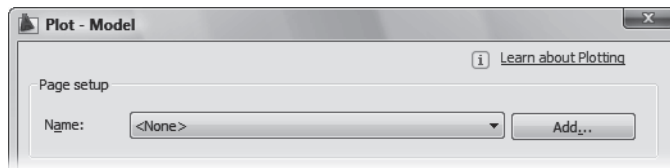
Note Use plot previews to discover plotting results before committing time, paper, and ink (toner) to a drawing that may be set up incorrectly. If the **Preview** button is grayed out, you have not yet selected a plotter for the layout.

Full Preview

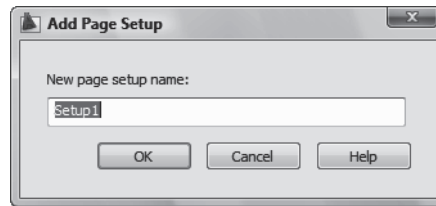
Click the **Preview** button to see the *full* preview, which shows the drawing as it would appear as a final plot on paper. See the `PREVIEW` command earlier in this chapter.

Save the Settings

If you plan to use the same plot parameters again, save them. In the Page Setup Name section (near the top of the Plot dialog box), choose the **Add** button.

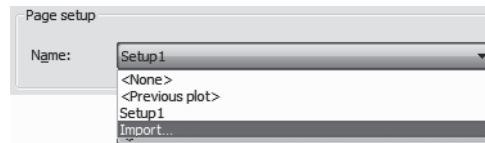


AutoCAD displays the User Defined Page Setups dialog box:



Enter a descriptive name, and then choose **OK**. The next time you plot this drawing, you can select the page setup name from the Page Setup | Name droplist, and all of the dialog box's parameters change to those of the saved page setup.

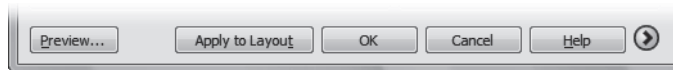
(As an alternative, you can select **Import** from the droplist.



AutoCAD displays the Select Page Setup From File dialog box; choose a .dwtg drawing file that you know contains page setups.)

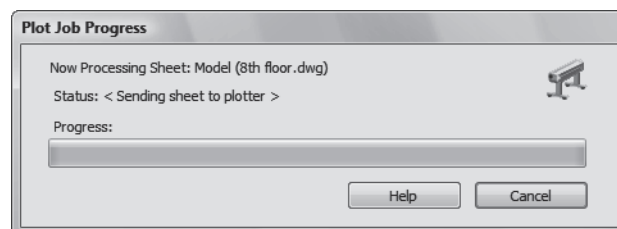
Plot the Drawing

When you have completed these steps, make sure the plotter is ready, and then choose the **OK** button at the bottom of the Plot dialog box.



(To save the settings but not plot the drawing, click **Apply to Layout**, and then **Cancel**.)

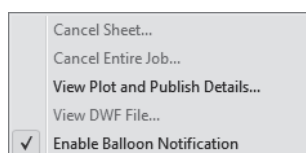
AutoCAD displays a dialog box indicating its progress plotting each layout.



In the tray at the right end of the status bar, look for the tiny pen plotter icon moving a sheet of paper back and forth – even though pen plotters hardly exist any more, notes the technical editor, kind of like the rarely-used-any-more diskette that inspired the icon for the Save button.



Right-click the plotter icon to see additional options:



Cancel Sheet — cancels the current sheet being plotted.

Cancel Entire Job — cancels multi-sheet plot jobs.

View Plot and Publish Details — displays the dialog box discussed in the next section.

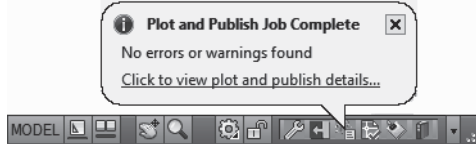
View DWF File — opens the .dxf file in the DWF Viewer, if drawings are published in DWF format.

Enable Balloon Notification — toggles the display of the balloon described below.

VIEWPLOTDETAILS

The VIEWPLOTDETAILS command reports on successful and failed plots.

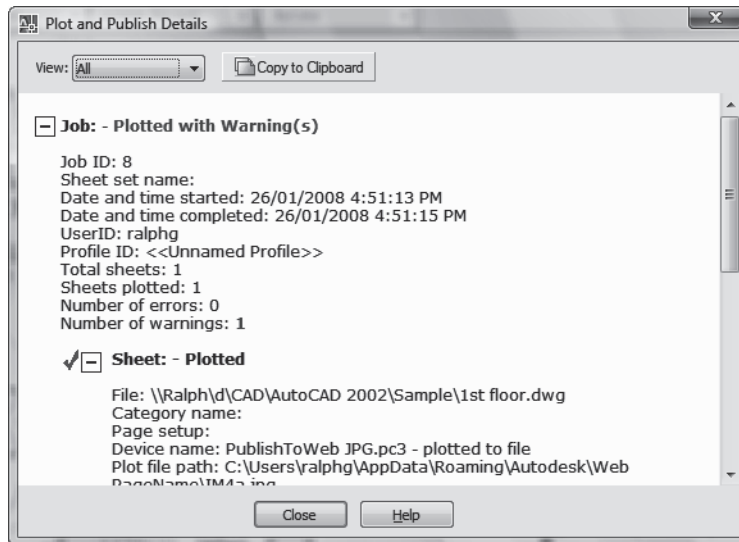
Once the plot is complete, AutoCAD displays a yellow balloon at the right end of the status bar. It reports whether (or not) the plot was successful.



Should the plotter be next to your desk, you don't need yellow balloons telling you that the drawing has been plotted. But when the plotter is located elsewhere, it's good to know the status without having to consume precious calories by getting up from your desk and walking over to it. ("This is so tempting," muses the copy editor. "Although," adds the technical editor, "it's the only exercise some people get.")

Click the blue underlined text to view a full report on the plot. AutoCAD displays the Plot and Publish Details dialog box.

(As alternatives, you can enter the VIEWPLOTDETAILS command, or right-click the plotter icon on the status bar, and then select **View Plot and Publish Details**. To dismiss the balloon, click the x in the upper right corner.)



The View droplist displays all reports or just those containing errors.

The **Copy to Clipboard** button copies the reports to the Clipboard; use the **Edit | Paste** command in a word processor to capture the report in a document.

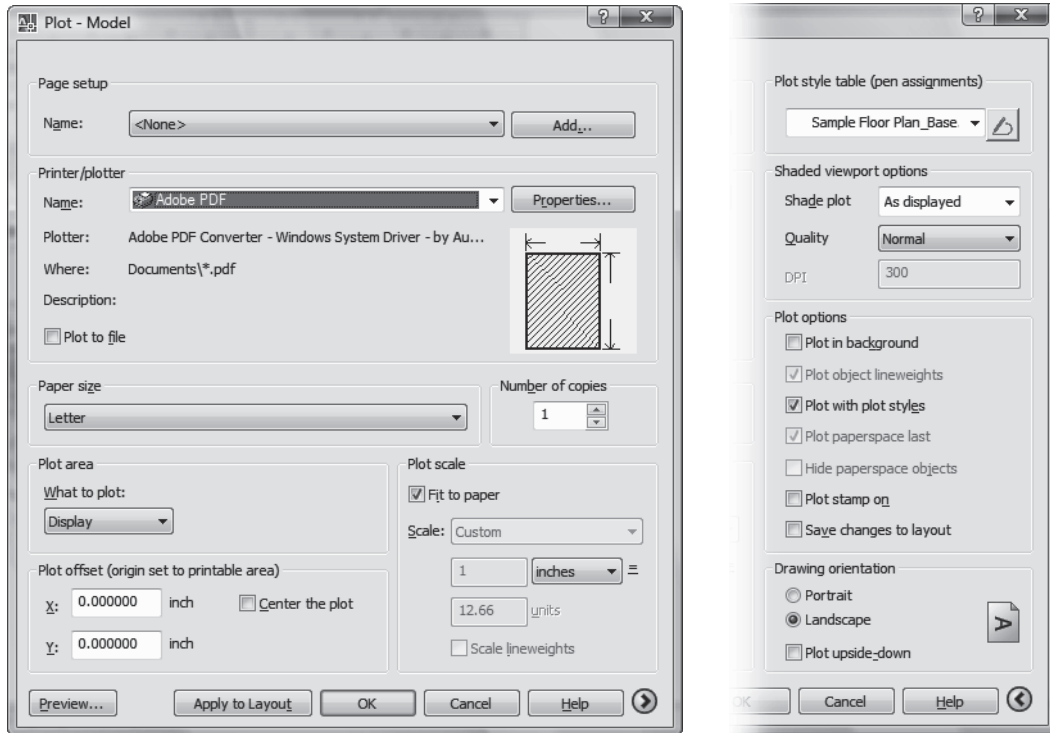
Click **Close** to close the dialog box.



PLOTTING DRAWINGS: ADDITIONAL METHODS

The **PLOT** command's dialog box sports a **More Options** button that expands to show more options. Click the button, or press **ALT+>**.

Let's look at each, except for **Plot Stamp On**, which is discussed later in this chapter.

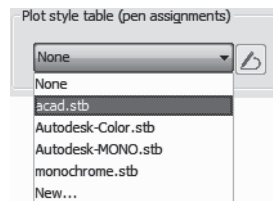


Select a Plot Style Table

The word “pen” in Pen Assignments derives from an earlier age when plotters plotted with actual pens — commonly ink pens, but also ballpoint pens, felt pens, and even pencils and knives (for cutting signs). The term has carried over to today's non-pen plotters that use laser and inkjet technology; now, it refers to varying widths, colors, and shades of gray.

Early releases of AutoCAD assigned colors of objects to pens. For example, all objects colored red (either by layer or by object) were plotted by a specific pen; those colored blue plotted with another pen, and so on. The pens didn't need to contain red or blue ink; typically they contained black ink and had tips of varying widths, such as 0.1" or 0.05".

Plot styles today assign plotter-specific properties to layers and objects, such as widths, colors, line-end capping, and patterns.

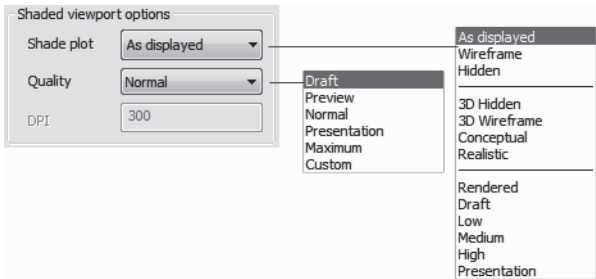


Select one of the preassigned plot style tables from the list. To create a new style, select **New** from the droplist.

To edit an existing style, click the **Edit**  button. You learn more about plot styles under the **STYLESMANAGER** command later in this chapter.

Select Viewport Shade Options

Shaded Viewport Options controls plotting of viewports. Viewports typically display drawings with 2D and 3D wireframe views, but sometimes you want 3D models plotted as renderings or with hidden-lines removed. This option lets you specify the type and quality of shading.



Although this option has “viewport” in its name, it also applies to drawings plotted in Model tab. You can check the effect of changing **Shade plot** in the preview window; **Quality** and **DPI** don’t show up until you plot on paper.

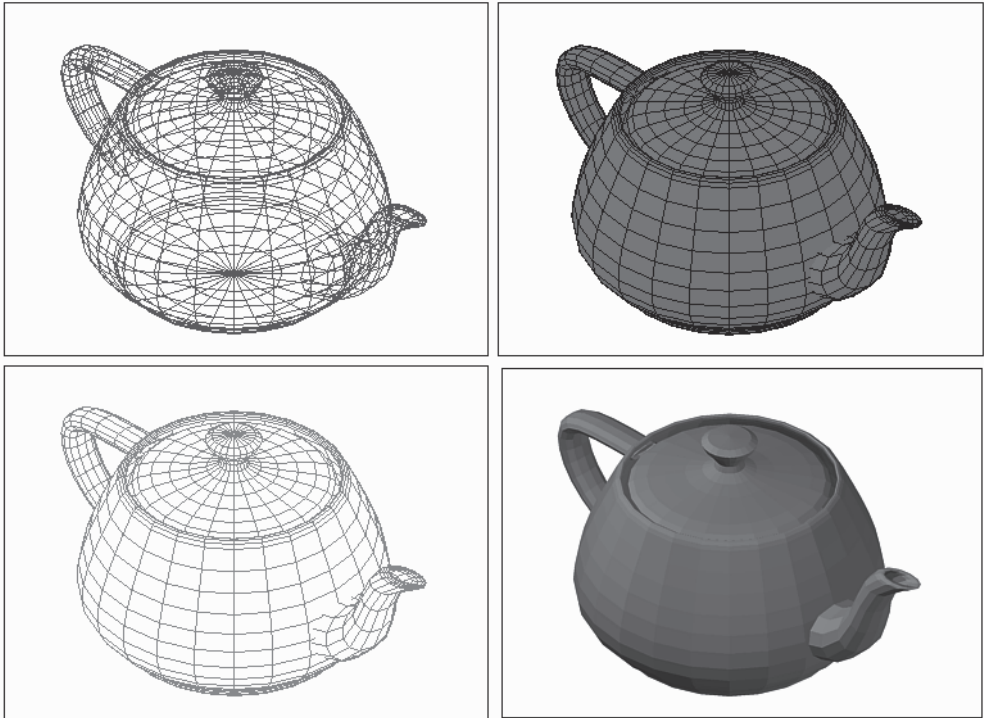
Under some conditions, the following options are unavailable:

Shade plot — unavailable when at least one viewport in layout mode has a Shade Plot other than “As Displayed” assigned to it by the Properties palette. This option is always available when plotting from the Model tab.

DPI — available only when **Quality** is set to “Custom.”

Shade Plot Options

The **Shade Plot** droplist applies visual and rendering styles, such as those applied to the samples illustrated below. Even if your graphics board cannot display all visual styles effects, such as shadows and materials, AutoCAD can make the plotter simulate the effects in the plotted output.



Top left: 3D drawing displayed in wireframe.
Bottom left: 3D drawing with hidden lines removed.

Top right: Gooch shaded (conceptual visual style).
Bottom right: 3D drawing rendered.



Note If a viewport does not plot, it may be that its display has been turned off. You can turn it back on with the **PROPERTIES** command: in the **Misc** section, change the **On** option to “Yes.”

Shade Plot	Comment
As Displayed	Plots objects as displayed in model or layout tabs. If shaded with the VISUALSTYLES command or rendered with the RENDER command, then the drawing is plotted that way.
Wireframe	Plots objects in wireframe, regardless of viewport display mode.
Hidden	Plots objects with hidden lines removed, regardless of display mode.
<i>Visual Styles</i>	
3D Hidden	Plots 3D objects with hidden lines removed.
3D Wireframe	Plots 3D objects in wireframe.
Conceptual	Plots 3D objects using Gooch rendering.
Realistic	Plots 3D objects in a realistic mode.
<i>If the drawing has other visual styles defined, they are included in this list.</i>	
<i>Rendered</i>	
Rendered	Plots 3D objects rendered, regardless of display mode.
Draft	Plots 3D objects at the lowest quality.
Low	Plots 3D objects...
Medium	at varying levels...
High	...of quality.
Presentation	Plots objects at the highest quality.
<i>If the drawing has other rendering modes defined, they are included in this list.</i>	

Quality and DPI Options

The **Quality** droplist specifies the resolution for shaded and rendered plots, because they are raster images. (Wireframe and hidden-line-removed are plotted as vectors.)

Quality	Meaning
Draft	Plotted at the output device’s lowest resolution.
Preview	Plotted at a maximum resolution of 150 dpi.
Normal	Plotted at a maximum of resolution 300 dpi.
Presentation	Plotted at a maximum resolution of 600 dpi.
Maximum	Plotted at the output device’s highest resolution.
Custom	Plotted at the resolution specified in the DPI text box

DPI is short for “dots per inch.”

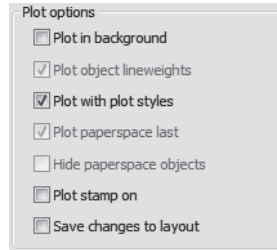


Note Two-dimensional drawings cannot be rendered, or have hidden lines removed or visual styles applied.

You can see the effect of shading and hidden-line removal on 3D drawings in the plot preview window. Note that applying visual styles and renderings to drawings takes longer, and can slow the preview display and increase the plotting time.

Plot Options

The **Plot Options** area lists miscellaneous options.



Plot in Background

When the **Plot in Background** option is turned on, AutoCAD plots the drawing “in the background,” allowing you to return to editing the drawing more quickly. (This is a replacement for autospooling, discussed later in this chapter.)

Always leave this option turned on, unless AutoCAD slows down unacceptably.

Plot Object Lineweights and Plot with Plot Styles

The **Plot with Lineweights** and **Plot with Plot Styles** options toggle each other; you can have one or the other, neither, but not both:

- **Plot with Lineweights** plots drawings with lineweights, *if* lineweights are turned on, and *if* lineweights have been assigned to layers and objects.
- **Plot with Plot Styles** plots drawings with plot styles, again only if defined and turned on.

Lineweights are described in Chapter 7, “Properties and Layers,” and plot styles are discussed later in this chapter.

Plot Paperspace Last

AutoCAD normally plots paper space objects before model space objects. Model space objects include viewport borders and title blocks. The **Plot Paperspace Last** option reverses the order.

Why does this matter? We’re not sure, but perhaps it has to do with how color printers lay down the ink. This option is not available when plotting from the model tab, because it has no paperspace elements.

The technical editor notes that the following options also affect the plotted output: there are differences between how Hidden modes hide circles with thickness (some look like a can, others like a pipe), and how values of the `DISPSILH` system variable create various looks.

Hide Paperspace Objects

The **Hide Paperspace Objects** option determines if hidden-line removal applies to 3D objects in paper space. This allows drawings not to show hidden-line removal, yet have hidden-line removed for the plot. The technical editor is unsure why anyone would want 3D objects in paper space, but here’s how to do it:

1. In model space, create a 3D solid model with internal details, such as slots and holes.
2. From the Edit menu, select **Copy**, and then copy the entire model.
3. Switch to a layout tab, and then paste the model (**Edit | Paste**).

4. Use the preview command to notice the differences:
 - **Hide Paperspace Objects** = off — solids in paper space appear as wireframes, showing internal details.
 - **Hide Paperspace Objects** = on — solids in paper space have hidden lines removed.

This option is overridden by the **Shade plot** option of viewports, and is not available when plotting from the model tab.

Plot Stamp On

The **Plot Stamp On** option toggles plot stamping on plotted drawings. See the PLOTSTAMP command later in this chapter.

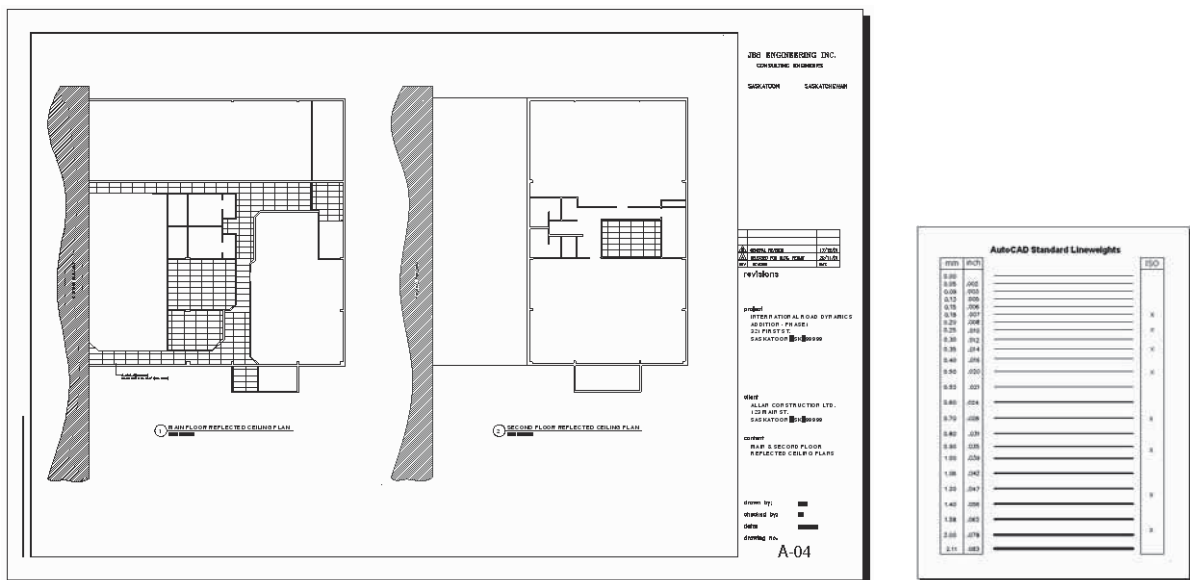
Save Changes to Layout

When turned on, the **Save Changes to Layout** option stores the changes made in this dialog box with the current layout.

The next time you plot from this layout tab, the same settings appear in the Plot dialog box. Select a different layout or drawing, and the settings are different.

Select the Orientation

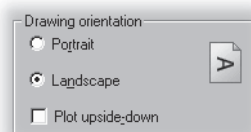
Larger CAD drawings are usually plotted in *landscape* mode, with the long edge of the page oriented horizontally. To do this, choose the **Landscape** button in the Drawing Orientation area.



Left: D-size drawing plotted in landscape mode.

Right: A-size drawing plotted in portrait mode.

Smaller drawings are often printed in *portrait* mode, where the long edge is upright. Choose the **Portrait** button.



The **Plot upside-down** option is handy if you stuck the paper with the title block the wrong way around in the printer.

AUTOSPOOL

AutoCAD has an “autospool” option that lets it *spool* plots as an alternative to background plots. Spooling is set up through the Add a Plotter wizard’s Port page.

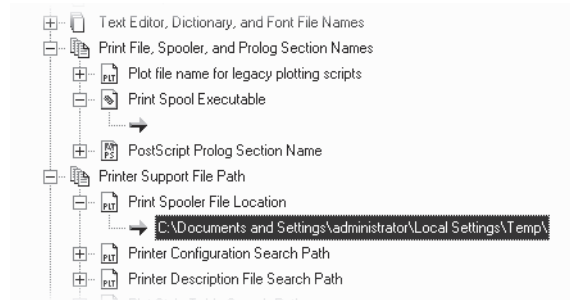
Spooling is a technique (short for “simultaneous peripheral operations online”) that speeds up printing jobs. When printing a document or plotting a drawing, the print data is sent to a file on disk; shortly thereafter, the spooling software starts up automatically, and sends the print data from the file to the printer. It is faster to save a file to disk than to print, so the application finishes the print job faster. (Spooling is also known as *buffering*, and in Windows it is handled by the Print Manager.)

AutoCAD allows you to use independent software for plot spooling. Before using the software, however, you must set up AutoCAD:

1. Install and configure the spooler software according to the vendor’s instructions.
2. AutoCAD needs to know the name of the spooler, as well as the folder in which to place the spool files.

From the Tools menu, select **Options**, and then the **Files** tab:

- Open **Print File, Spooler, and Prolog Section Names**, and in **Print Spool Executable**, specify the spooler program name.
- Open **Printer Support File Path**, and in **Print Spool File Location**, specify the folder name.



Choose **OK**.

3. Set up the plotter by selecting **File | Plotter Manager**, and then working through the wizard.
4. Start the **PLOT** command, and then select the plotter from the Plotter Configuration droplist.

OPTIONS

The **OPTIONS** command lets you set defaults for plotting. These are set in two of the dialog box's tabs: the **Plot and Publish** tab, and the **Files** tab.

PLOT AND PUBLISH TAB

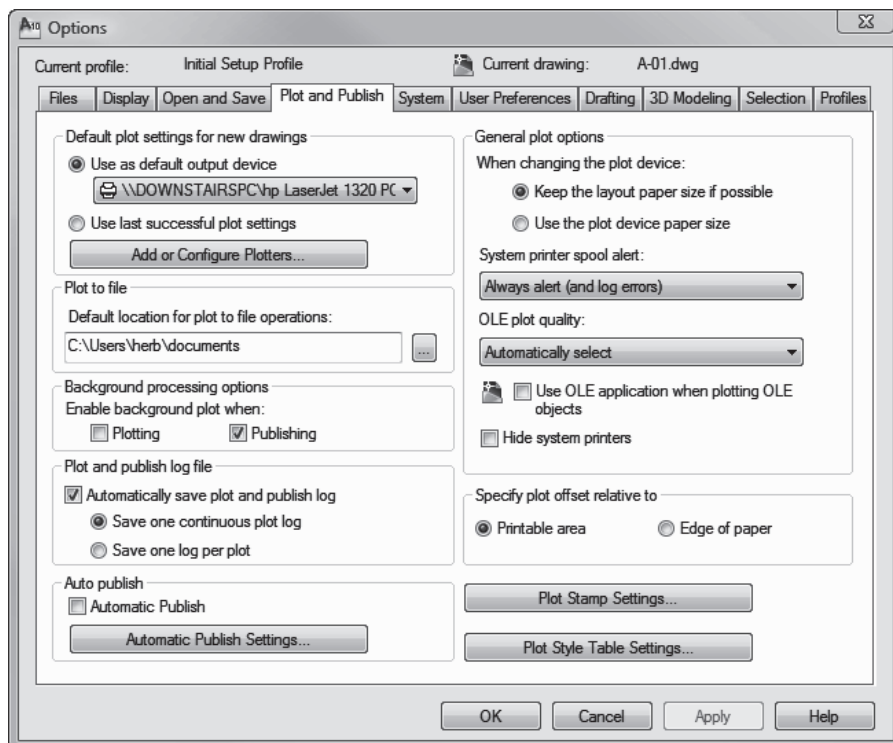
The **Plot and Publish** tab contains defaults specific to plotting and publishing drawings. “Plotting” means printing one drawing at a time, while “publishing” means printing sets of drawings (two or more).

Default Plot Settings for New Drawings

Here you specify the default plotter:

- ☉ **Use As Default Output Device** selects the default printer from a list of system printers and devices defined by AutoCAD's .pc3 (printer configuration) files. It will be used for new drawings, as well as for those saved in versions prior to AutoCAD 2000.
- **Use Last Successful Plot Settings** selects the last-used plotter settings, instead of the default printer.

Click the **Add or Configure Plotters** button to display the **Plotter Manager** window; see the **PLOTTERMANAGER** command later in this chapter.



Plot To File

Default Location for Plot to File Operations specifies the drive and folder for storing drawings plotted to files.

Type the path, or click the **...** button to display the **Select Default Location for All Plot-to-File Operations** dialog box.

Background Processing Options

Background plotting lets you work on drawings while AutoCAD generates the plot. When **PLOT**, **-PLOT**, **PUBLISH**, and **-PUBLISH** are used in scripts, this setting is ignored and drawings are plotted and published in the foreground.

- ☐ **Plotting** performs background plot jobs; see the **PLOT** command in this chapter.
- ☒ **Publishing** performs background publish jobs; see the **PUBLISH** command.

Plot and Publish Log File

The log file saves plotting and publishing results in a CSV text file (comma-separated value) that can be imported into spreadsheet and database programs. Data include Job ID, Job name, Sheet set name, Category name, Date and time started and completed, Sheet name, Full file path, Selected layout name, Page setup name, Named page setup path, Device name, Paper size name, and Final status.

- ☒ **Automatically Save Plot and Publish Log** creates log files:
 - ☒ **Save One Continuous Plot Log** stores all plot data in a single log file.
 - ☐ **Save One Log File Per Plot** stores data about each plot in a separate log file.

AutoPublish

AutoCAD can save the drawing in DWF or PDF formats at the same time as it is saved in DWG format. Turn on this option when you work with these electronic plot files, and need them to be up to date. Otherwise, leave this option turned off, because it prolongs the save-file process. (PDF support is [new to AutoCAD 2010](#).)

- ☐ **Automatic Publish** saves drawings in DWF or PDF format during the **SAVE** and **CLOSE** commands. (You can also toggle auto-DWF/PDF publishing with the **AUTOPUBLISH** command.)

Click the **Automatic Publish Settings** button to specify whether drawings are saved in DWF, DWFx, or PDF format.

General Plot Options

When changing the plot device:

- ☒ **Keep the Layout Paper Size If Possible** uses the paper size specified by the plot setup. When the printer cannot handle the specified paper size, AutoCAD displays warning message, and then uses the size of paper specified by the PC3 plotter configuration file, or by the system printer's settings.
- ☐ **Use the Plot Device Paper Size** ignores the size specified by the layout, and uses the paper size specified by the plotter's PC3 configuration file or the default system settings.

System Printer Spool Alert determines the type of alert displayed by port conflicts during plot spooling:

- **Always Alert (And Log Errors)** displays an alert and records the error to the log file.
- **Alert First Time Only (And Log Errors)** displays an alert the first time only, and then records the error to the log file.
- **Never Alert (And Log First Error)** never displays alerts, but records error to log files.
- **Never Alert (Do Not Log Errors)** never displays alerts and does not record to log files.

OLE Plot Quality specifies the quality of plotted OLE objects:

- **Monochrome** is suitable for black text.
- **Low Graphics** is suitable for colored text and simple graphics.
- **High Graphics** is meant for photographs.
- **Automatically Select** allows AutoCAD to choose the setting.

☐ **Use OLE Application When Plotting OLE Objects** launches the application that created the OLE object in order to improve the plot quality.

☐ **Hide System Printers** toggles the display of Windows system printers. System printers configured by the Add-a-Plotter wizard are always displayed in the Plot dialog box.

Specify Plot Offset Relative To

- ☒ **Printable Area** offsets the plot relative to the printable area.
- ☐ **Edge of Paper** offsets the plot relative to the edge of the paper.

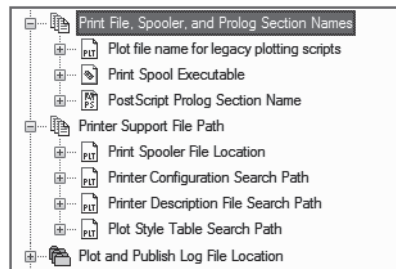
Other Options

Plot Stamp Settings button opens the Plot Stamp dialog box; see the PLOTSTAMP command.

Plot Style Table Settings button opens the Plot Style Table Settings dialog box; see the STYLESMANAGER command.

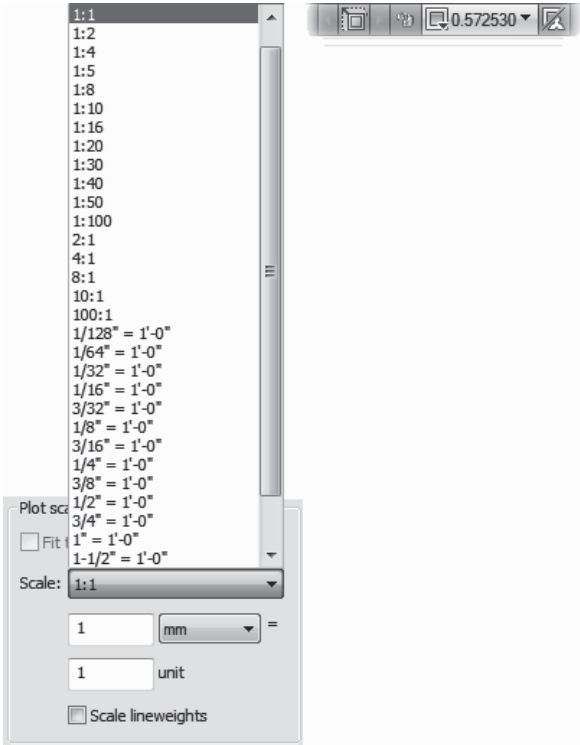
FILE TAB

The Options dialog box's File tab specifies the default location for plot-related files, as illustrated below:



SCALELISTEDIT

The **SCALELISTEDIT** command customizes lists of plot scales. The command also controls the list of scale factors displayed for viewports, page layouts, and annotative scales. Use this command to shorten the list of unused scale factors and to add custom scales.

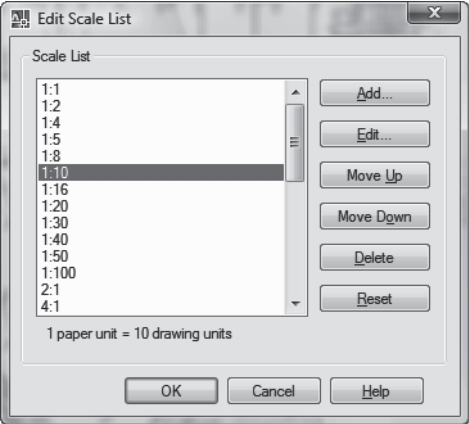


Left: Scale factors listed in Plot dialog box.
Right: The same scale factors available in the drawing bar.

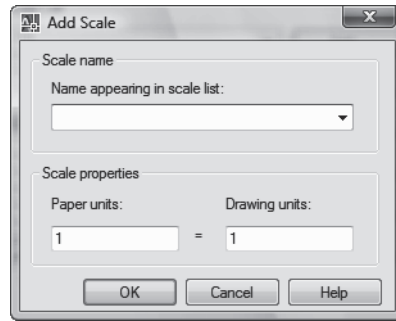
- I. To edit scale factors, start the **SCALELISTEDIT** command:
 - From the Format menu, choose **Scale List**.
 - In the ribbon's Annotate tab, choose **Scale List** from the Annotation Scaling panel.
 - At the 'Command:' prompt, enter the **scaledlistedit** command.

Command: **scaledlistedit** (Press ENTER.)

AutoCAD displays the Edit Scale List dialog box, which lists the scale factors currently defined.



- To add a new scale factor, click **Add**; to edit existing scale factors, select one, and then click **Edit**.



Name Appearing in Scale List — provides descriptive names that appear in place of scale factors; alternatively, enter an actual scale factor, such as 1:10.

Paper Units — specifies the size of objects on paper. This value is usually 1 (one) for drawings larger than the paper.

Drawing Units — specifies the size of objects in the drawing. This value is usually 1 for drawings smaller than the paper.

- The Move Up and Move Down buttons move selected scale factors up and down the list, which allows you to place the frequently-used scales at the top of the list.
- The **Delete** button removes the selected scale, while the **Reset** button deletes all custom scales, and then restores the default list.
- Click **OK** to exit the dialog box.

The `-SCALELISTEDIT` command performs the same functions at the command line:

Command: **-scalelistedit**

Enter option [?/Add/Delete/Reset/Exit] <Add>: *(Enter an option.)*

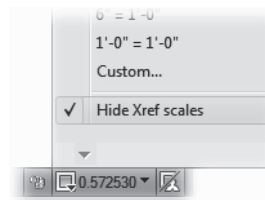
HIDEXREFSCALES

The `HIDEXREFSCALES` system variable usefully hides all scale factors imported from xref'd drawings when set to 1:

Command: **hidexrefscases**

Enter new value for `HIDEXREFSCALES` <0>: **1**

To view imported scale factors, change this system variable to 0. Alternatively, uncheck the option in the drawing bar:

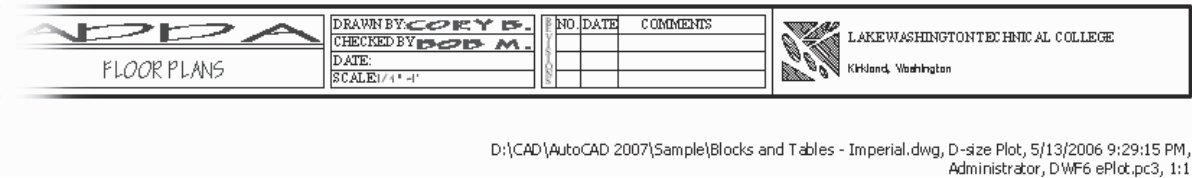


PLOTSTAMP

The PLOTSTAMP command stamps plots with information about the drawing and plot.

When you plot sets of drawings, it sometimes becomes difficult to determine which set was plotted most recently, or which drawing files produced the plots. To identify the plots more readily, apply *plot stamps*.

A plot stamp consists of one or two lines of text that list the drawing name, date and time plotted, the plotter device, plot scale, name of the computer that generated the plot, and so on. In addition, AutoCAD lets you specify two custom pieces of data.



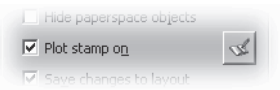
Plot stamp (below the title block) that provides information about the drawing and the plot.

Plot stamps are often placed along the edge of the drawing, as illustrated above. You can change the location, as well as the font and size of text. The plot stamp parameters can be saved to files, which is useful for billing clients, seeing who is hogging the plotter, and so on.

You access plot stamps with the PLOTSTAMP command, or from the **Plot Stamp Settings** button of the Plot and Publish dialog boxes.

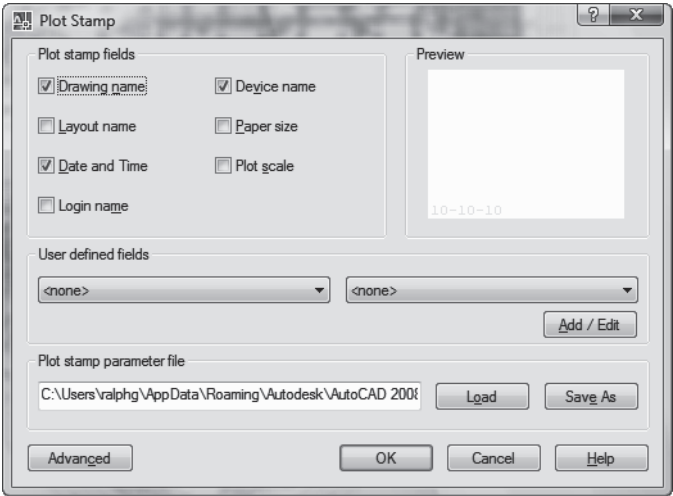
TUTORIAL: STAMPING PLOTS

- I. To apply a stamp to the plotted drawing, start the **PLOTSTAMP** command:
 - In the Plot or Publish dialog boxes, choose the **Plot Stamp Settings** button.



- At the 'Command:' prompt, enter the **plotstamp** command.
Command: **plotstamp** (Press ENTER.)
- Alternatively, enter the **ddplotstamp** (the command's old name) alias at the 'Command:' prompt.

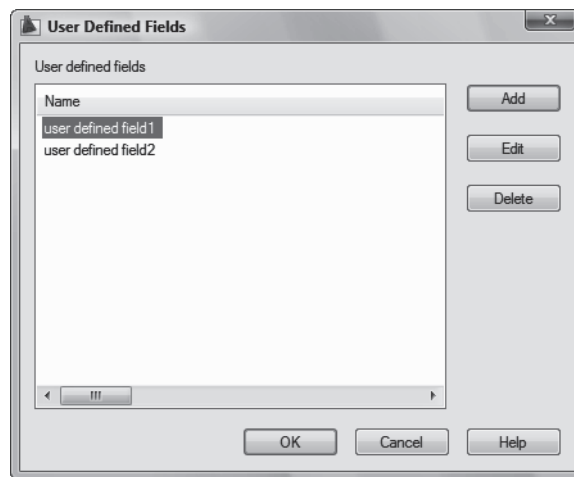
In all cases, AutoCAD displays the Plot Stamp dialog box.



2. In the Plot Stamp Fields area, select the text you wish stamped on the plot:

Plot Stamp Fields	Example	Comments
Drawing Name	C:\17_22.DWG	Full path and file name of the drawing.
Layout Name	Model	Layout name; "Model" if plotted from model space.
Date and Time	6/26/2003 10:24:42 AM	Date and time of the plot; format is determined by the Regional Settings dialog box of the Control Panel.
Login Name	Administrator	Windows login name, as stored in the LOGINNAME sysvar.
Device Name	Lexmark Optra R+	Name of the plotting device.
Paper Size	Letter 8 1/2 x 11 in	Size of the paper.
Plot Scale	1:0.8543125	Plot scale factor; when you see an unusual scale factor, such as that shown in the example, this means the drawing was probably scaled to fit the margins.

3. To define your own fields, click the **Add/Edit** button. AutoCAD displays the User Defined Fields dialog box.



Click **Add**, and then enter text. You can add as many user-defined fields as you wish, but AutoCAD specifies a maximum of two per plot stamp. When done, click **OK**.

To include a user-defined field with the plot stamp, select them from the droplist under User Defined Fields.

4. Click **Save As** to save the settings to file. This allows you to reuse the settings with other drawings, or swap with friends.

AutoCAD displays the Plotstamp Parameter File Name dialog box. Enter a file name, and then click **Save**. AutoCAD saves the data in a .pss (plot stamp parameter) file.

5. Click **OK**.

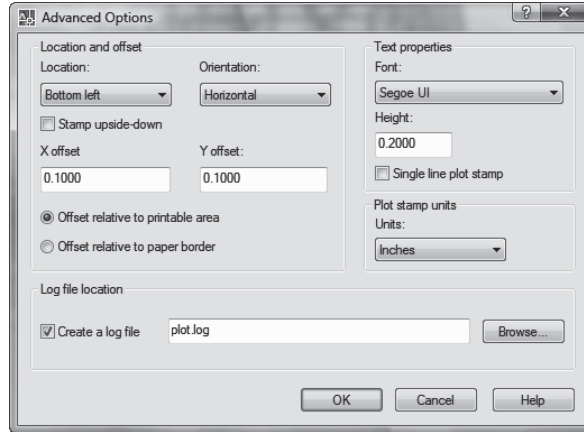


Notes When the options of the Plot Stamp dialog box are grayed out, this means that the *inches.pss* or *mm.pss* file in AutoCAD's \support folder is set to read-only. To change the setting with Windows Explorer: (1) right-click the .pss file, (2) select **Properties**, (3) uncheck **Read-only**, and (4) click **OK**.

The plot stamp data is not saved with the drawing, but is generated anew with each plot. Plot stamps are plotted with color 7 (on raster plotters) or pen 7 (on pen plotters).

STAMPING PLOTS: ADVANCED OPTIONS

The Advanced Options dialog box provides additional options for locating the stamp on the plot. In the Plot Stamp dialog box, click **Advanced**.



The Location and Offset area positions the stamp on the plot. The preview, unfortunately, is not located in this dialog box, so you need to go through a cycle: change settings, click **OK** to get out of this dialog box, check the plot preview, and return.

The Stamp Upside-down option is useful, because otherwise the stamp can be confused for other text in the drawing. Also, some filing systems work better with upside-down plot stamps.



Note A large offset value may position the plot stamp text beyond the plotter's printable area, which may cause the text to be cut off. To prevent this, use the **Offset relative to printable area** option.

The Text Properties section selects the font and size of text. I recommend a narrow font, such as Arial Narrow or Future Condensed, to ensure all the plot stamp text fits the page. The default size, 0.2", tends to be too large; I recommend 0.1" or smaller.

Plot Stamp Units defines the offset and height numbers used by this dialog box. Select from inches, millimeters, and pixels.

The Log File Location area lets you specify the name and folder for the plot logging file. The .log file is in ASCII format, and contains exactly the same information as stamped on the plot, such as:

C:\17_22.DWG,Model,6/26/2003 12:34:48 PM,Administrator,Lexmark Optra R plus.

Click **OK** to exit the dialog box.

PAGESETUP

The **PAGESETUP** command assigns plotters to layouts.

AutoCAD automatically assigns the default Windows system printer to new drawings. Right before plotting the drawing, you can select another printer in the **PLOT** command's dialog box. Or, you can assign a specific printer, such as your office's networked large-format plotter, as part of the initial setup for template drawings. This is done with the **PAGESETUP** command.

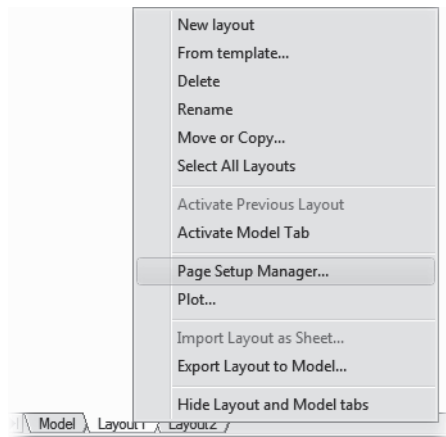
Page setups are lists of instructions that tell AutoCAD how to plot pages: model space, layouts, and sheets. (I think Autodesk uses the word “page” because AutoCAD prints these onto *pages* of paper.) The flexibility of page setups means that you can have a different page setup for every layout or sheet in a drawing. Each could be plotted by a different plotter on different size paper, and at different scales and orientations — in color or in monochrome. Page setup data are stored in the *.dwg* file.

The Page Setup Manager lists the layouts to which the page setup will be applied. This list can consist of a single layout name, or of all the layouts in the current drawing. (When opened from the Sheet Set Manager, it displays the name of the current sheet set.)

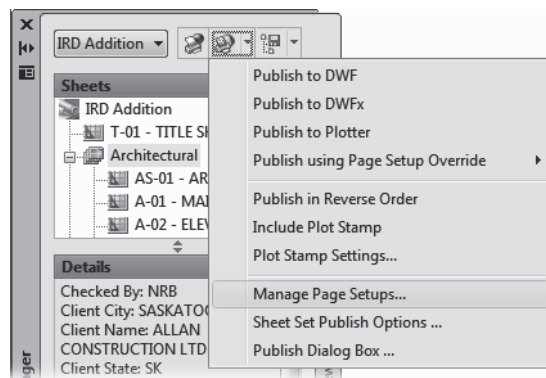
Page setups can be accessed in a number of places: in the Plot dialog box, from the Sheet Set Manager palette, and from the layout tabs.

TUTORIAL: PREPARING LAYOUTS FOR PLOTTING

- I. To prepare drawings for plotting, start the **PAGESETUP** command:
 - From the ribbon's Output tab, choose **Page Setup** from the Plot panel.
 - Right-click a layout tab, and then select **Page Setup Manager** from the shortcut menu.



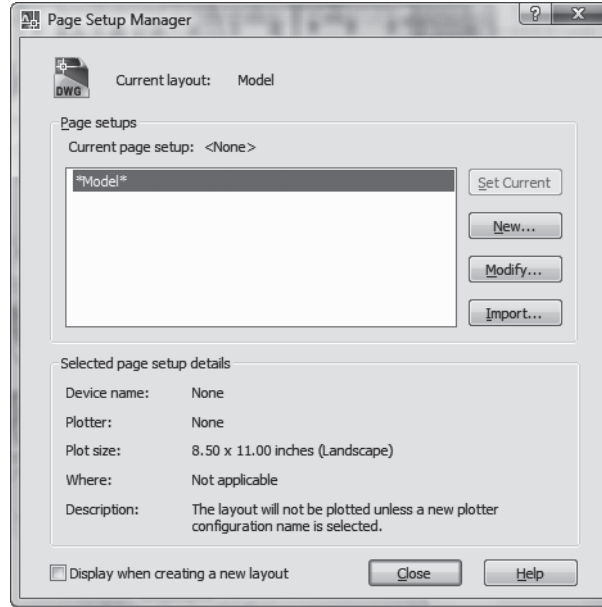
- In the Sheet Set Manager, click the **Publish** button, and then select **Manage Page Setups** from the menu.



- At the 'Command:' prompt, enter the **pagesetup** command.

Command: **pagesetup** (Press ENTER.)

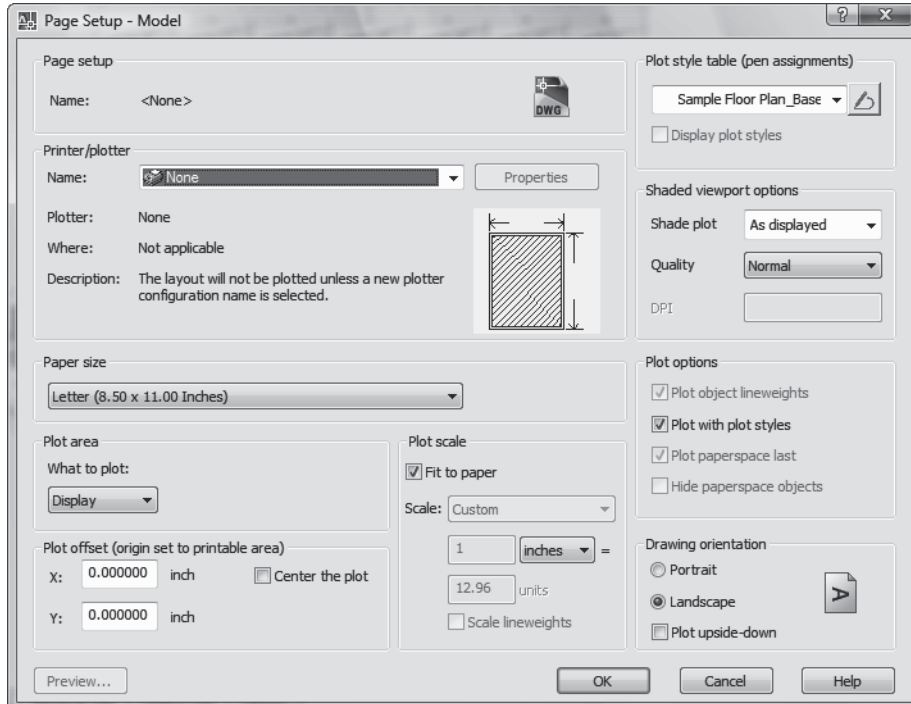
In all cases, AutoCAD displays the Page Setup Manager, which lists page setups, if any are already defined.



(The **Display When Creating a New Layout** option displays this dialog box each time you create a new layout tab.)

2. Look in the lower half of the dialog box. Notice that no plotter has been selected for this layout. To specify a plotter, click **Modify**.

Notice the Page Setup dialog box, which looks almost identical to the Plot dialog box.



3. In this dialog box, you can preset many options for future plotting. The most important, however, is specifying the printer or plotter to use.
In the Printer/Plotter area, select a printer from the Name droplist.
The names of printers in the droplist vary with every computer, and include (1) Windows printer drivers, (2) AutoCAD plotter drivers, and (3) attached network printer drivers.
4. Set other parameters, if you wish. If you are unsure which settings to use, I recommend these:

Plot Area	Extents
Plot Offset	Center the Plot
Plot Scale	Fit to Paper
5. Click **OK**. Your drawing is now ready to plot.



PUBLISH

The **PUBLISH** command plots *set of drawing*: one or more drawings and/or combinations of model and layout tabs. (In contrast, the **PLOT** command plots just one drawing at a time.) A drawing set consists of one or more drawings and layouts that are plotted, in order, at one time — these are called “sheets” by Autodesk.

Engineering and architectural offices often use drawing sets to combine all drawings belonging to a project. This makes it easier to plot many drawings at once, such as at specific stages of large projects. Alternatively, drawing sets can be generated as *.dxf* or *.pdf* files for archiving or plotting later.

This command selects drawing files and layouts, and then lets you reorder, rename, and copy them. The command also simply plots drawings — in effect, it is a Super Plot command.

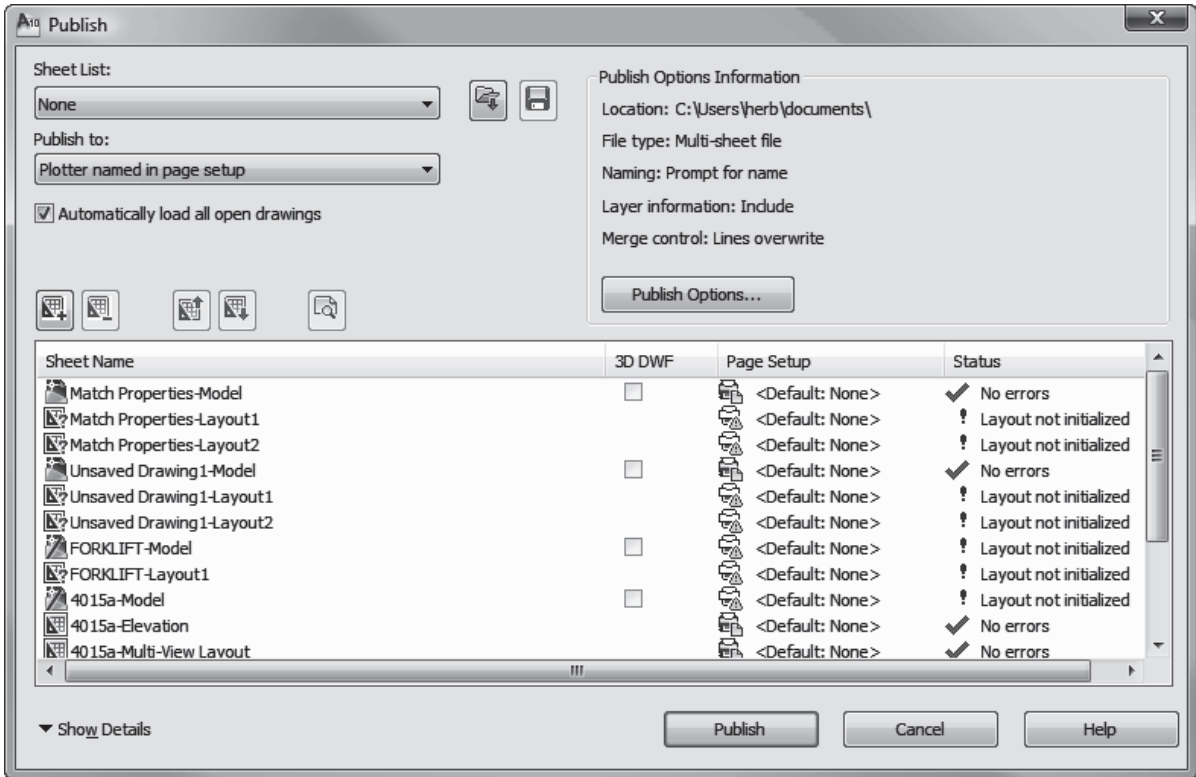
Lists of drawing sets can be saved as *.dsd* files (drawing set description) for later reuse. (The **BATCHPLT** utility has been removed from AutoCAD; its functions are now performed by the **PUBLISH** command.)

TUTORIAL: PUBLISHING DRAWING SETS

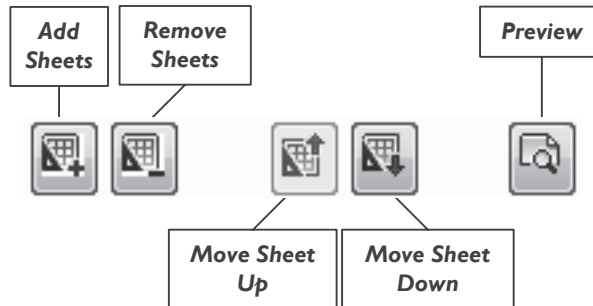
1. To create a drawing set, start the **PUBLISH** command:
 - In the ribbon's Output tab, choose the **Publish** button from the Publish panel.
 - Hold down the **CTRL** key, select two or more layout tabs, right-click, and then choose **Publish Selected Layouts** from the shortcut menu.
 - At the 'Command:' prompt, enter the **publish** command.

Command: **publish** (Press ENTER.)

In all cases, AutoCAD displays the Publish Drawing Sheets dialog box. Initially, the dialog box lists the names of the current drawing(s) and layout(s) open in AutoCAD.



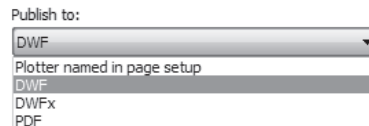
Note Buttons with icons have these meanings:



2. AutoCAD provides a number of ways to create lists of drawings (sheets) to print (plot):

- a. **Automatically Load All Open Drawings** option adds all open drawings and layouts to the list of sheet names. If you don't want AutoCAD to do this for you, turn off the option (new to AutoCAD 2010).
- b. Click the curiously-named **Add Sheets**  button to add drawing files from your computer's hard drive or network locations. You can choose among DWG, DWX, DWT, DXF, and PDF format in model or layouts view or both. As AutoCAD adds drawings to the list, it also extracts the layouts. Each file name is listed separately, with one of the following suffixes:
 - “**-Model**” — indicates the model tab (model space view).
 - “**-Layoutn**” or “*LayoutName*” — indicates the layout tab number or name. (If sheets of the same name already exist in the list, AutoCAD will prompt you to change their names.)

- c. Click the **Load Sheet List**  button to bulk-load drawings through DSD (drawing set descriptions) or BP3 (batch plot v3) files. DSD files are created by the **PUBLISH** command; the files simply contains the list of drawings you want to publish. Typically, this is a list of project drawings that you may need to print on a regular basis. (BP3 files were created by the Batch Plot utility program provided with AutoCAD 2000 through 2004.)
- d. If there are some sheets you want to remove from the list, select them, and then click the **Remove** button.
 - Hold down the **SHIFT** key to select a range of layout names.
 - Hold down the **CTRL** key to select nonconsecutive layout names.
 AutoCAD removes the selected layouts without asking if you're sure — even if you're not.
3. Drawing sets are usually plotted in a specific order. Use the **Move Up** and **Move Down** buttons to change the order of the layouts. (You can reverse the plot order, so that the last layout is plotted first. Some plotters stack output pages in reverse order; this option fixes the problem. To do so, click **Show Details**, and then click the **Reverse Plot Order** button.)
4. The drawing sets can be “published” to the printer, or saved as an electronic plot format for transmittal by email. Make your selection in the Publish To section among DWF, DWFx, PDF, or another format set up with the **PAGESETUP** command:



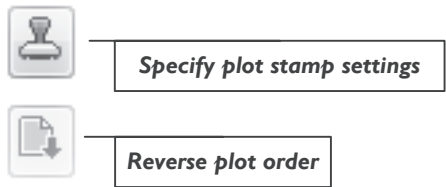
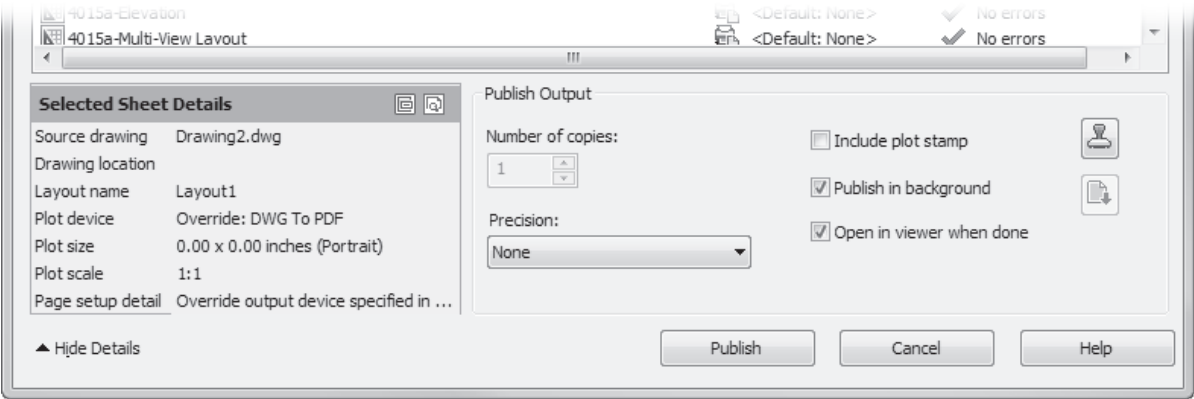
- **Plotter Named in Page Setups** sends all of the layouts to a plotter — but the layouts might not end up being plotted by the printer you expect. Prior to pressing the **Publish** button, ensure each layout is set up for the correct printer. This complexity is necessary, because drawing sets sometimes need to be plotted to a variety of printers.
 - **3D DWF** saves the model sheets in 3D .dwf format.
5. Click **Save List** to save the list of layouts for later reuse.
 6. Click **Publish**. AutoCAD loads each drawing, and then generates the plot or the .dwg file. A dialog box notes its progress.

If a drawing or layout cannot be found, AutoCAD skips it and carries on publishing the next layout. If errors are found, AutoCAD generates a .csv (comma separated value) log file, which you can read with a spreadsheet program.

When done, AutoCAD asks if you wish to view the output in an external viewer — DWF or DWFx in Design Review (provided with AutoCAD) or PDF in Acrobat Reader (free from www.adobe.com/acrobat).

SHOW DETAILS

The Publish dialog box places some commands in the Details section; you see this section after clicking the **Show Details** button.



Publish Output

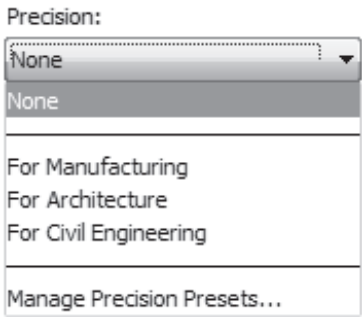
The **Publish Output** section contains additional options for generating sheets.

Number of Copies

You can have the plotter output more than one copy of the sheets; this option is unavailable when outputting to PDF or DWF files, because they can print multiple copies from their viewer programs.

 Precision

The **Precision** droplist selects a predefined level of precision, the units and subunits, and resolution of non-vector entities (such as gradients) for DWF or PDF formats. Choose one of the presets (For Manufacturing, and so on), or choose **Manage Precision Presets** to edit them.



Plot Stamps

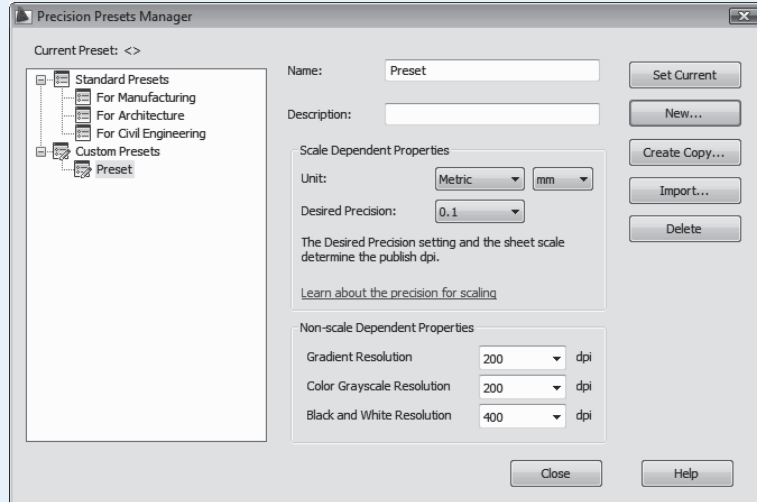
Include Plot Stamp adds plot stamp data to the edge of each plot.

The  **Plot Stamp Settings** button displays the Plot Stamp dialog box for customizing the plot stamp information; see the PLOTSTAMP command elsewhere in this chapter.



PRECISION PRESETS MANAGER DIALOG BOX

This dialog box can be accessed through the **PUBLISH** command's **Precision** option, as well as through the **EXPORTSETTINGS** command's Export to DWF/PDF Options dialog box's **Override Precision** option.



Current Preset specifies the name of the current precision preset.

Name names the preset.

Description describes the purpose of the preset.

Scale Dependent Properties options

Unit chooses Metric or Imperial units, and subunits (mm, m, feet, inches, etc).

Desired Precision specifies the precision at which the drawing will be exported.

Non-scale Dependent Properties

A lower resolution setting reduces file size but makes the gradients look posterized and images look grainier.

Gradient Resolution specifies the resolution of gradient hatch areas.

Color Grayscale Resolution specifies the resolution of images that are color or grayscale.

Black and White Resolution specifies the resolution of monochrome images.

Set Current sets the selected preset as the working preset.

New creates new presets; displays the New Precision Preset dialog box which prompts for a name and description.

Create Copy copies of the selected preset; displays the Copy Precision Preset dialog box, which prompts you for a name and description.

Import imports .dsd Drawing Set Description files created by the **PUBLISH** command; displays the Select List of Sheets dialog box.

Delete erases the preset.

The  **Plot Stamp Settings** button displays the Plot Stamp dialog box for customizing the plot stamp information; see the PLOTSTAMP command elsewhere in this chapter.

Other Options

Publish in Background returns to the drawing editor more quickly by generating the sheets while you return to editing.

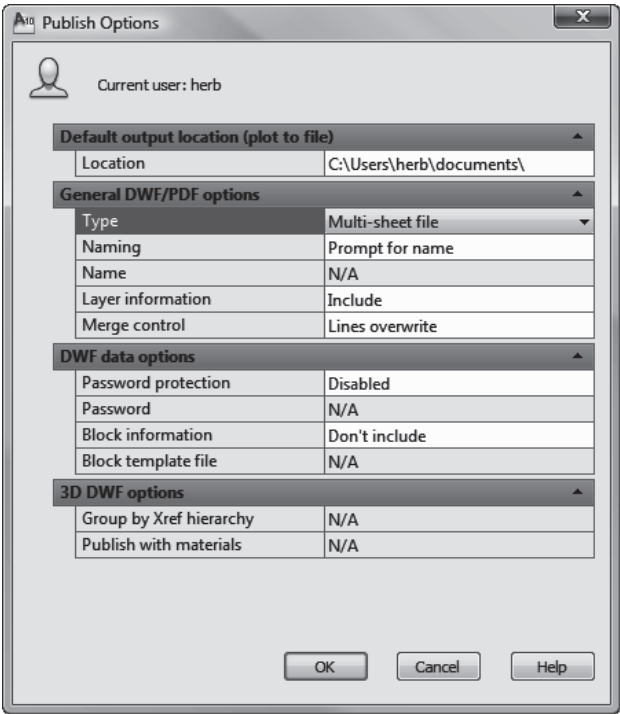
Plot Order

Currently publishing in default order  button to reverse the order in which layouts are plotted. AutoCAD normally publishes the drawing set in the order listed by the dialog box. Some plotters output pages face-up, which means the set ends up in reverse order; this button fixes the problem.

Open in Viewer When Done displays DWF files in Design Review or PDF files in Acrobat Viewer v7 or later (new to AutoCAD 2010).

PUBLISH OPTIONS

Before publishing the drawing list in DWF format, click the **Publish Options** button. The Publish Options dialog box controls output for DWF and PDF files. (This dialog box was redesigned in AutoCAD 2010.)



Current User

The name that appears next to the icon in the upper-left corner varies. When this dialog box is accessed from the **PUBLISH** command, you see the Windows userid of the currently logged-in user, such as “Administrator”; when accessed from the Sheet Set Manager palette, you see the sheet set’s name.



Left: Current user name when dialog box accessed through **PUBLISH** command.
Right: Current sheet set name when dialog box accessed through **SHEETSET** command.

Default Output Location

The **Default Output Location** (formerly named Directory) section determines where plot files are saved when you publish drawing sheets. This option is available only when saving in DWF or PDF format and when plotting to file.

Enter a different drive and folder name, or click the ... button to display the Select Folder for Generated Files dialog box.

General DWF/PDF Options

The General DWF/PDF Options section specifies how drawings are published as PDF and DWF files.

Type

AutoCAD can publish all sheets in a single file, or each sheet in its own file:

Single-Sheet File — each sheet generates its own file. Use this option for compatibility with old viewer software that does not accept electronic files in multisheet format.

Multi-Sheet File — all sheets are stored in a single file. Use this option for the convenience of publishing to a single file.

Naming

When AutoCAD creates a single-sheet DWF, the *.dwf* file takes on the drawing's (sheet's) name. But what name should AutoCAD use when publishing more than one drawing? Here you specify how the file is named:

Specify Name — you specify the path and file name right here.

Prompt for Name — you specify the path and file name later, after you click the **Publish** button.

Name

The Name option specifies the name of the file (valid for the Specify Name option only).

Layer Information

This option determines whether layers are included in the file. When turned off, no layer names are included; this can be useful for limiting the amount of information provided to clients.

Merge Control

When lines and other objects cross each other in the drawing, AutoCAD can output the intersections by merging (lines of color blend) or overwriting (last line drawn is drawn overtop other lines).

DWF Data Options

The following options apply to DWF files only.

Password Protection

DWF files are often distributed by email or on CD; adding a password prevents unauthorized people from viewing the files. The recipients must have the password to open the file; how they receive the password is up to you. If you forget the password, the DWF file cannot be recovered, although you can generate another DWF file from the original drawings.

Disabled — password protection is turned off.

Specify Password — allows you to specify the password here.

Prompt for Password — AutoCAD prompts you for the password later when you click the **Publish** button.

Password

The **Password** option specifies the password (when Specify Password option is enabled). You can use letters, numbers, punctuation, and non-ASCII characters for the password. The password is *case-sensitive*, which means that “autocad” is not the same password as “AutoCAD.”

Block Information

This option determines how blocks are exported:

- Include** — includes blocks and attribute data in the file.
- Don't Include** — explodes blocks to remove block and attribute information.

Block Template File

You can use a DXE or BLK block template file to specify which blocks and attributes should be included in the file.

- Create** — creates a new *.blk* block template file; displays the Publish Block Template dialog box.
- Edit** — edits existing *.blk* files; displays the Publish Block Template dialog box.
- ...** — opens the Select Block Template file.

3D DWF Options

Usually, the **PUBLISH** command generates 2D DWF files. It can output 3D ones under two conditions: (1) when “3D DWF” is selected from the Publish dialog box’s Page Setup/3D DWF column, and (2) when the drawing is in model space only. (AutoCAD does not generate 3D DWFs from layouts, because they are 2D only.)

Once 3D DWF is selected for a Model sheet, the Publish Options dialog box makes the 3D DWF Options available:

3D DWF options	
Group by Xref hierarchy	No
Publish with materials	Yes

- Group by Xref Hierarchy** — lists objects under each xref name, if externally-referenced drawings are linked to the sheet.
- Publish with Materials** — includes material with objects; materials are used by Design Review.



Note The technical editor suggests setting up the DWF plotter with a very large paper size. He finds that this dramatically increases the resolution at the cost of a slight increase in file size.

PLOTTERMANAGER

The **PLOTTERMANAGER** command creates and edits plotter configurations.

AutoCAD plots to any printer connected to your computer. This includes printers found in your office, large-format plotters used in engineering and architectural offices, and files on disk. (Large-format plotters create D- and E-size plots that are roughly three to four feet, or one meter, across.)

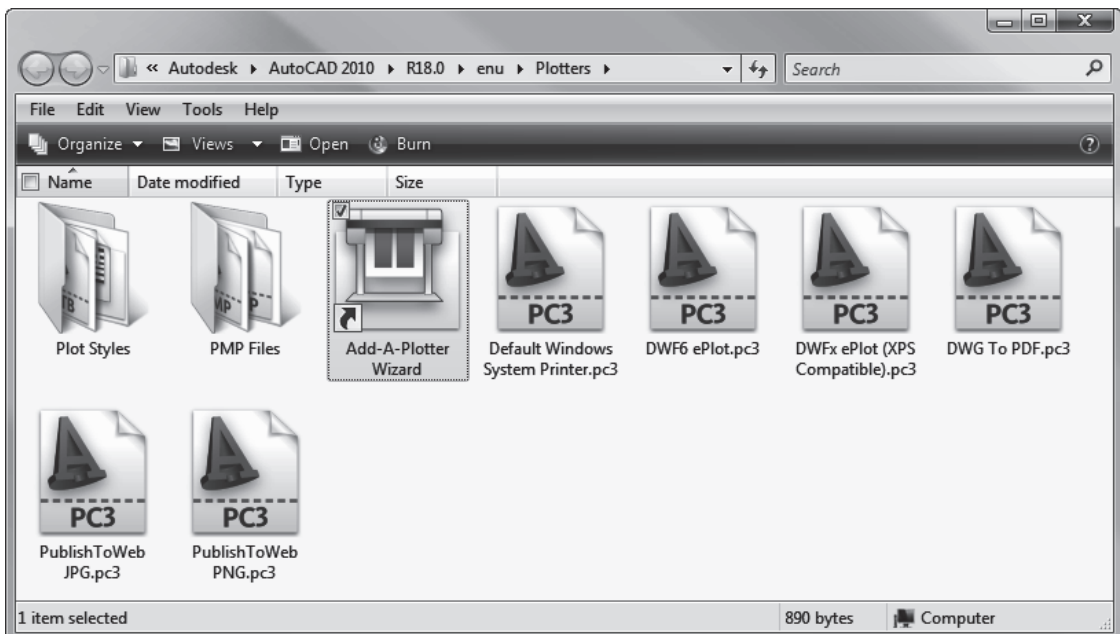
Most software relies on Windows to provide device drivers. The problem is that the device drivers provided by Microsoft are not accurate and flexible enough for AutoCAD's high-precision plotting. For this reason, Autodesk includes its own set of drivers, known as HDI.

To plot drawings with HDI drivers, first run the Plotter Manager. This configures in great detail how the plotters should print the drawings.

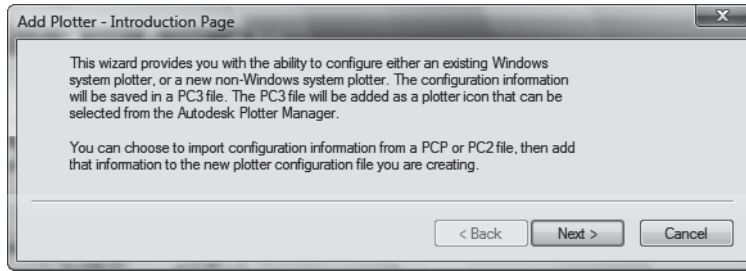
TUTORIAL: CREATING PLOTTER CONFIGURATIONS

- To create configurations for plotters, start the **PLOTTERMANAGER** command:
 - From the 2D ribbon's Output tab, select **Plotter Manager** from the Plot tab.
 - At the 'Command:' prompt, enter the **plottermanager** command.

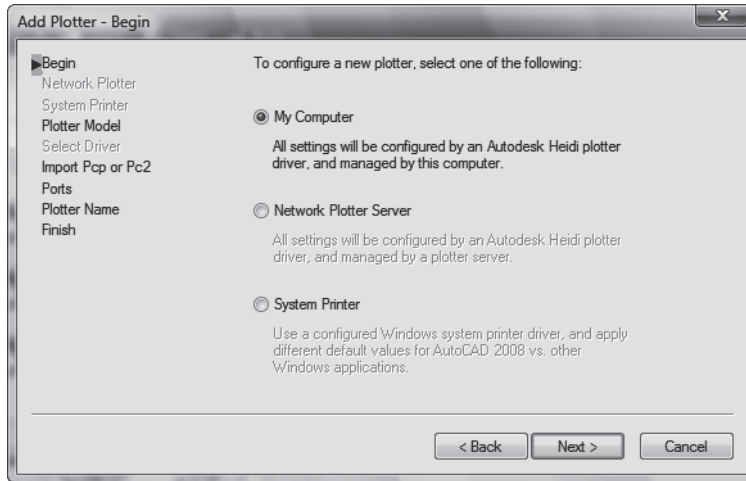
Command: **plottermanager** (Press ENTER.)
- AutoCAD displays the contents of the `C:\Users\username\AppData\Roaming\Autodesk\AutoCAD 2010\R18.0\enu\Plotters` folder. Select a PC3 file to edit (consulting the "Tutorial: Editing Plotter Configurations" that follows), or double-click **Add-a-Plotter Wizard** to create a new plotter definition.



- When you choose Add-a-Plotter Wizard, AutoCAD starts the *addplwiz.exe* program, which displays the Introduction Page.



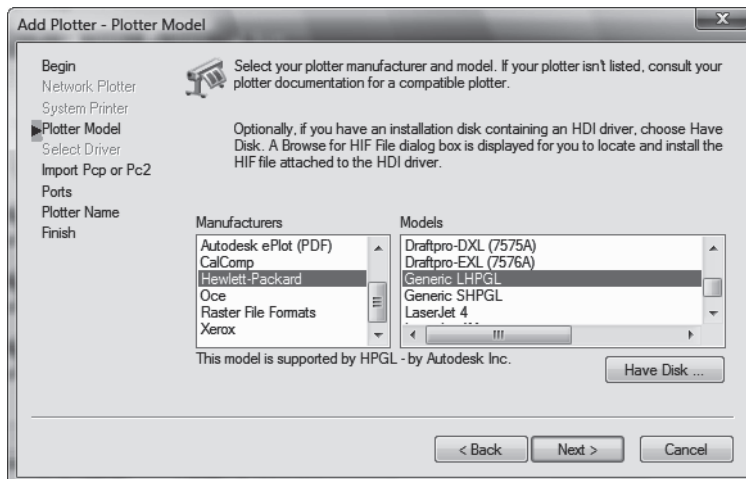
- Choose **Next**.
- In the Begin page, set up AutoCAD with one of these styles of printer:



- **My Computer** are local printers and plotters controlled by AutoCAD's plotter drivers.
- **Network Plotter Server** are printers located on the network, also controlled by AutoCAD's plotter drivers.
- **System Printer** are local and network printers controlled by Windows printer drivers.

Unless you have a reason to do otherwise, choose **My Computer**, and then click **Next**.

- AutoCAD displays a list of printer and plotter drivers provided by Autodesk.



From the Manufacturers list, select the brand name of the plotter.

From the Models list, select the specific model number.

If your new plotter comes with AutoCAD-specific drivers on a diskette or CD, choose the **Have Disk** button.

(If you had selected Network Plotter Server, AutoCAD would have prompted you for the network location of the server. If System Printer, AutoCAD would have prompted you to select one.)

Choose **Next**.

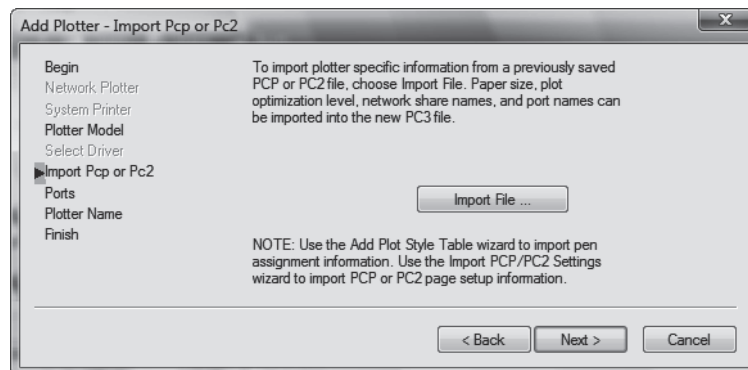


Notes If you do not see the name of your plotter's manufacturer, check the documentation. Often it lists brand names and model numbers of compatible plotters. If you cannot find this information, try selecting Adobe for PostScript printers and Hewlett-Packard for large-format inkjet plotters.

To plot the drawing in a raster format, select **Raster File Formats**, and then a specific format:

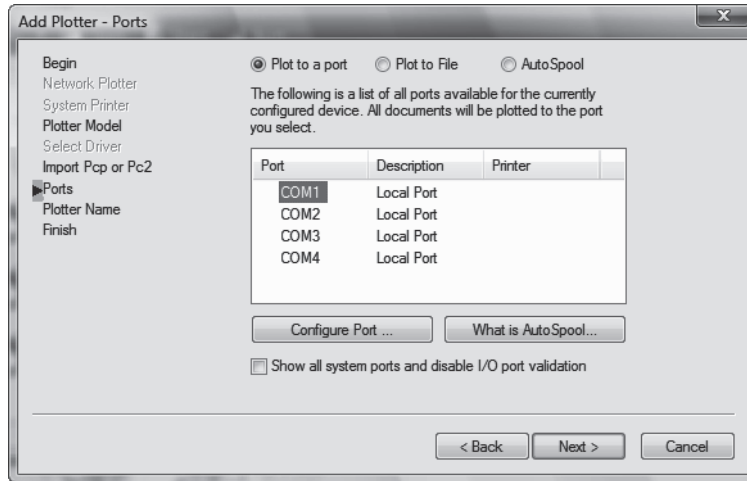
Format	Color Depths	File Extension
Independent JPEG Group JFIF	Gray, RGB	.jpg
Portable Network Graphics PNG	Bitonal, gray, indexed, RGB, RGBA	.png
TIFF Uncompressed	Bitonal, indexed, gray, RGB, RGBA	.tif
TIFF Compressed	Bitonal, indexed, gray, RGB, RGBA	.tif
CALS MIL-R-28002A Type 1	Bitonal	.cal
Dimensional CALS Type 1	Bitonal	.cal
MS-Windows BMP Uncompressed	Bitonal, gray, indexed, RGB	.bmp
TrueVision TGA 2.0 Uncompressed	Indexed, gray, RGB, RGBA	.tga
Z-Soft PC Paintbrush PCX	Indexed, RGB	.pcx

7. The Import PCP or PC2 page is important only if you created plotter configuration files with AutoCAD Release 13 (.pcp) and 14 (.pc2). Here you import these files for use with newer releases of AutoCAD (as .pc3 files).



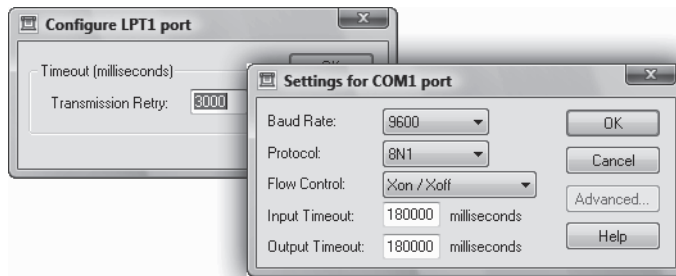
Choose **Next**.

8. The Ports page can be a difficult step for some users. Here you select the *port* to which the plotter is connected — or no port at all.



Your choices are:

- **Plot to a Port** — AutoCAD sends the drawing to the plotter through a local port or network port, such as a parallel port (designated LPT), serial port (designated COM), or USB port. Select this option for drawings plotted by printers and plotters.
 - **Plot to File** — AutoCAD sends the drawing to a file on disk. Select this option to save the drawing to disk as a file in both plotter and raster formats.
 - **AutoSpool** — AutoCAD sends the drawing to a file in a specified folder (defined by AutoCAD's Options dialog box), where another program processes the file. Select this option only if you know what you are doing. (If you need to ask, then you don't know.)
9. Some ports have further options. If necessary, choose the **Configure Port** button:

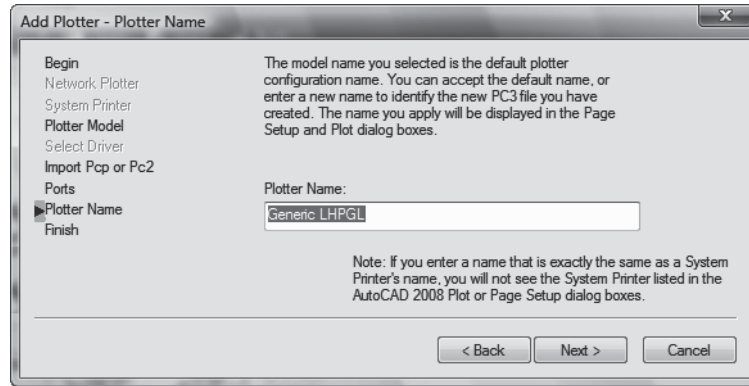


- **Serial** ports specify communications settings.
- **Parallel** and **USB** ports specify the transmission retry time.
- **Network** ports specify nothing, curiously enough.

Change settings, and then click **OK**.

Choose **Next**.

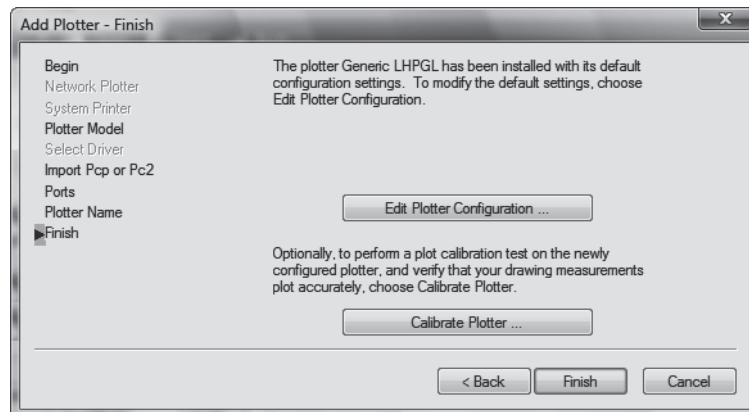
10. In the Plotter Name page, give the plotter configuration a name.



Enter a descriptive name, and then choose **Next**.

11. The Finish page has three buttons:

- **Edit Plotter Configuration** — Displays AutoCAD's Plotter Configuration Editor dialog box, which allows you to specify options, such as media source, type of paper, type of graphics, and initialization strings.
- **Calibrate Plotter** — calibrates the plotter. This allows you to confirm, for example, that a ten-inch line is indeed plotted 10.0000000 inches long.
- **Finish** — completes the plotter configuration.



12. Choose **Finish**.

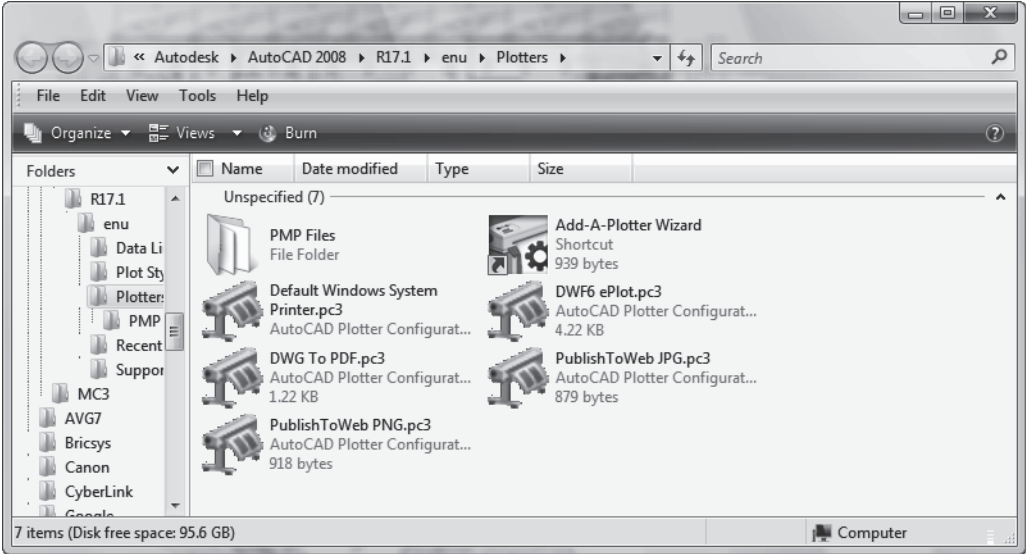
This configures the new plotter for AutoCAD. When you next use the **PLOT** command, this configuration appears in the list of available plotters.

TUTORIAL: EDITING PLOTTER CONFIGURATIONS

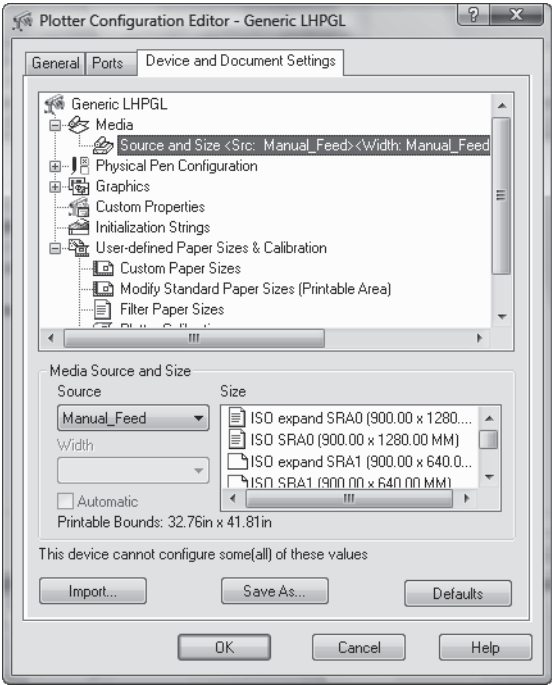
- 1. To edit a plotter configuration, start the **PLOTTERMANAGER** command:
 - From the 2D ribbon's Output tab, select **Plotter Manager** from the Plot tab.
 - At the 'Command:' prompt, enter the **plottermanager** command.

Command: **plottermanager** (Press ENTER.)

Again, AutoCAD requests that Windows display the Plotters window, which lists all HDI plotter configurations as .pc3 files.



- 2. Double-click any .pc3 icon, except *Default Windows System Printer.pc3*, because it must be edited through Windows.
- 3. Windows displays the Plotter Configuration Editor dialog box. Click the **Device and Settings** tab.



The available settings vary, depending on the capabilities of the plotter. (Shown below are settings for the HP LaserJet 5SiMX printer.) Some settings are unavailable when the plotter does not support them. Other settings are handled through the **Custom Properties** button.

Selected options are noted in angle brackets, <like this>. When a change is made to a setting, AutoCAD places a red checkmark in front of it.



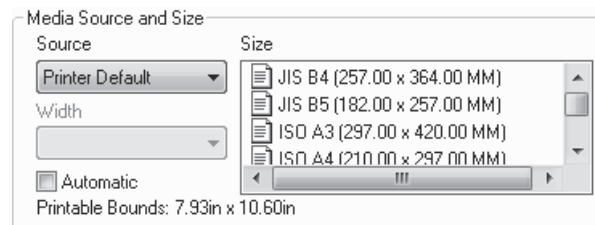
4. Change the settings, and then press **OK**.

THE PLOTTER CONFIGURATION EDITOR OPTIONS

The Plotter Configuration Editor lists features and capabilities that AutoCAD takes advantage of. Not all settings listed below are available for all printers and plotters. Some features must be set through Windows: select the **Custom Properties** node, and then click the **Custom Properties** button.

Media Source and Size

The **Media Source and Size** option specifies the paper source and size and type of paper.



Source specifies the location of the paper. Examples include trays, sheet feed, and roll feed.

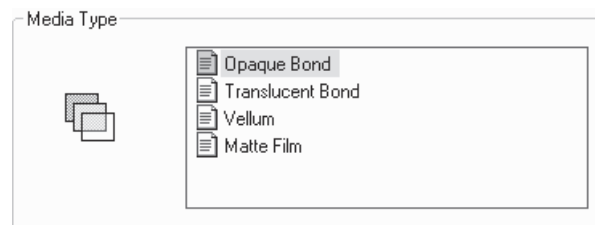
Width specifies the width of the paper; applies only to roll-fed sources of paper.

☐ **Automatic** means that the printer determines the paper source.

Size lists the standard and custom paper sizes the printer is capable of handling.

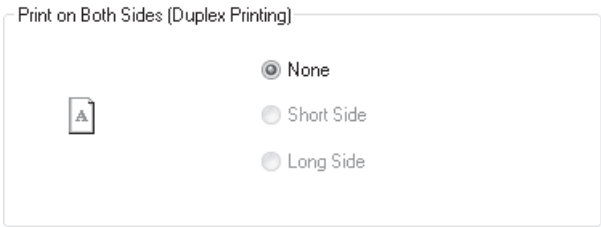
Media Type

The **Media Type** option determines the type of the paper, since this can affect how the plotter lays down its ink — not all printers support this.



Duplex Printing

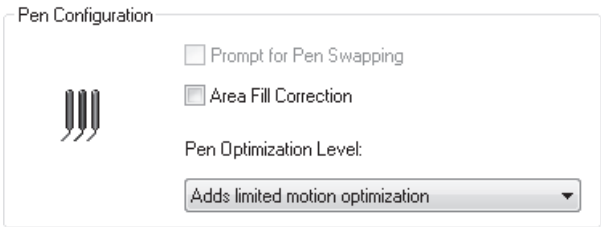
The **Duplex Printing** option determines whether the paper is printed on both sides — if the printer is capable of it.



- Ⓐ **None** turns off double-sided printing.
- Ⓑ **Short Side** sets the binding margin on the short edge.
- Ⓒ **Long Side** sets the binding margin on the long edge.

Physical Pen Configuration

The **Physical Pen Configuration** option is for pen plotters only; it is largely obsolete, or nearly so. It controls the actions of each pen.



- Ⓐ **Prompt for Pen Swapping** forces AutoCAD to pause the plot so that you can change pens; available for single-pen plotters only.
- Ⓑ **Area Fill Correction** forces AutoCAD to plot filled areas by half-a-pen width narrower, so as not to draw filled areas too wide.

Pen Optimization Level optimizes pen motion to reduce total plotting time. This setting is not effective with very slow computers and slow plotters.

Physical Pen Characteristics

Physical Pen Characteristics allows you to specify the color, width, and manufacturer-recommended top speed of each pen (in inches or millimeters per second).

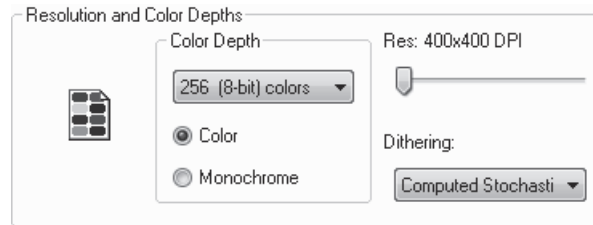
Physical Pen Characteristics				
Pen No.	Color	Speed (mm./s)	Width (mm)	
1	■ Black	20.00	0.25	⌵
2	■ Black	20.00	0.25	
3	■ Black	20.00	0.25	
4	■ Black	20.00	0.25	
5	■ Black	20.00	0.25	

Graphics

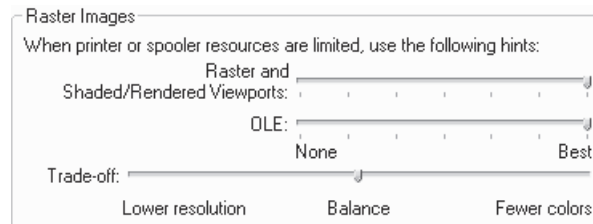
The **Graphics** options specify options for vector and raster graphics and for TrueType fonts. The settings here vary depending on the capabilities of each printer and plotter.

Resolution and Color Depths / Vector Graphics

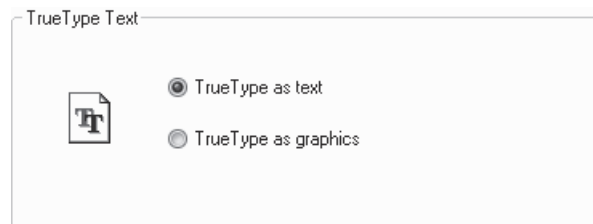
The **Resolution and Color Depths** section (a.k.a. Vector Graphics) specifies options for *color depth* (number of colors), resolution (dpi), and *dithering* (simulation of a color by mixing two colors). Some printers trade off fewer colors for higher resolution.



The **Raster Images** option trades off plotting speed for output quality (for plotters without pens).



The **TrueType** option specifies how TrueType fonts are handled — available only with the PostScript Level 3 driver.



- **TrueType as Text** — characters sent as text; this is more efficient, but it results in the Courier font being substituted for fonts to which the printer has no access.
- **TrueType as Graphics** — characters are sent as graphical images; this is less efficient, but guarantees that the fonts will look correct.

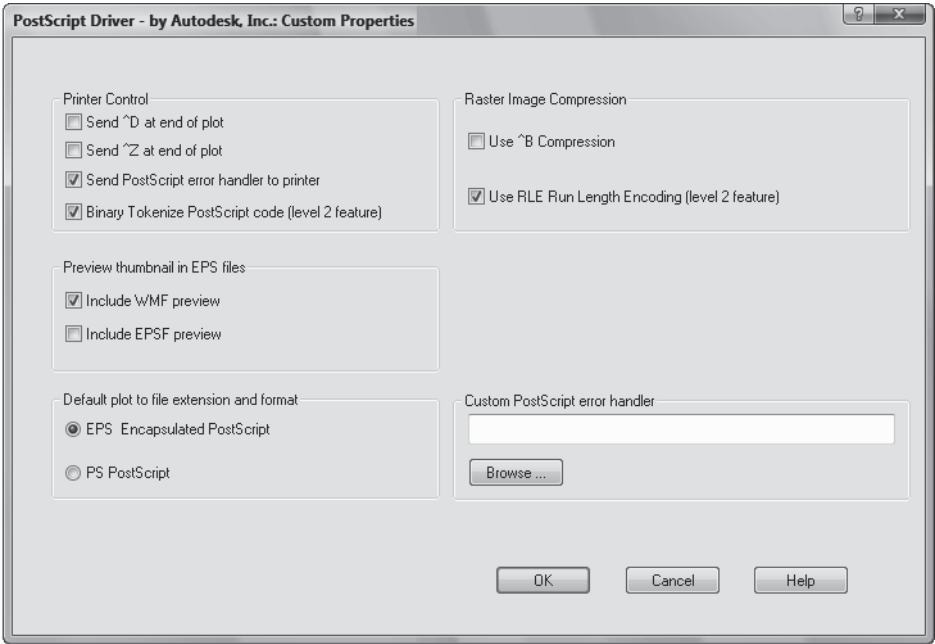
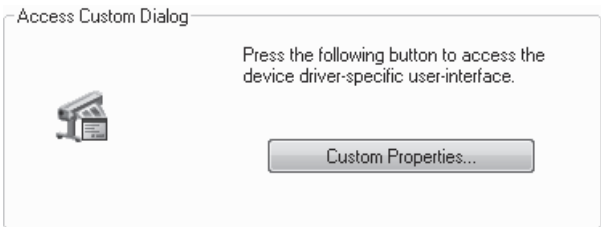
The **Merge Control** option specifies the look of crossing lines. This option is not valid when printing all colors as black, when using PostScript printers, and when the printer does not support merge control.



- **Lines Overwrite** — only topmost lines are visible at intersections.
- **Lines Merge** — colors of crossing lines are merged.

Custom Properties

The **Custom Properties** option displays the Custom Properties button. Click the button to view the dialog boxes that control the printer.



Initialization Strings

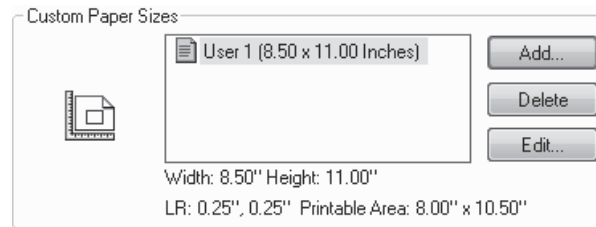
The **Initialization Strings** option sets control codes for non-system printers, including those for pre-initialization, post-initialization, and termination. These codes allow you to use plotters and printers not supported entirely by AutoCAD.



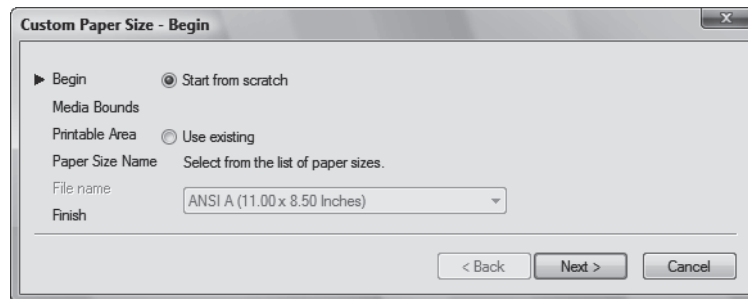
Note Use the backslash (\) to emulate escape strings. For example, \27 is sent as the escape character, and \10 as the line-feed character.

User-Defined Paper Sizes and Calibration

The **User-Defined Paper Sizes and Calibration** option allows you to specify custom paper sizes (those not recognized by AutoCAD) and to calibrate the printer.



To help you define custom paper sizes, AutoCAD runs the Custom Paper Size wizard. Click **New** to start it.



Note Calibration ensures drawings are plotted accurately. Autodesk recommends, “If your plotter provides a calibration utility, it is recommended that you use it instead of the AutoCAD utility.”

Following printer calibration, AutoCAD stores the calibration data in a *.pmp* (plotter model parameter) file. The *.pmp* file must be attached to the printer’s *.pc3* file, unless it was created during the Calibration stage of the Add-a-Plotter wizard.

Save As

To save the settings, click the **Save As** button. You can share the *.pc3* file with other AutoCAD users through the **Import** button.

STYLESMANAGER

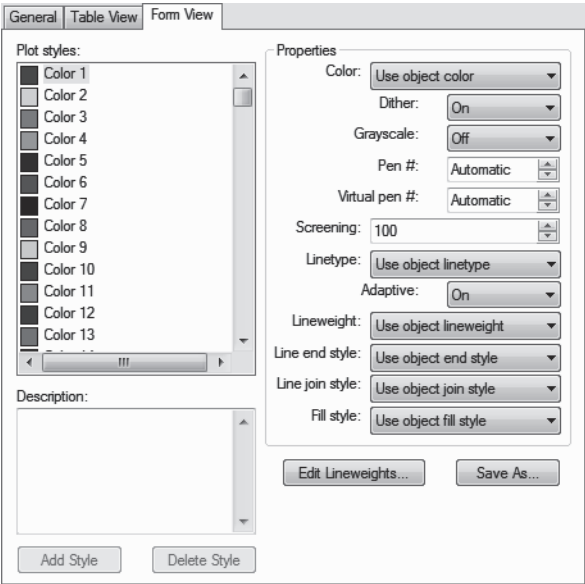
The STYLESMANAGER command creates and edits plot style tables.

Plot styles control the printing properties of objects. Depending on the capabilities of the printer, plot styles control color or grayscale, dithering and screening, pen number and virtual pens, linetype and lineweight, line end and join styles, and fill style. Plot style *tables* are collections of plot styles assigned to layouts and the model tab.

AutoCAD works with two types of plot style table: color-dependent and named.

COLOR-DEPENDENT PLOT STYLE TABLES (.CTB FILES)

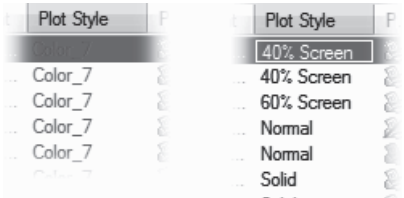
Color-dependent plot style tables (.ctb files) assign plot styles by the color of the layer or the objects, if ByLayer color has been overridden. (Prior to AutoCAD 2000, this was the only way to control printing. During the old PLOT command, the CAD operator assigned AutoCAD colors to pen numbers. For example, blue may have been assigned pen #3. During plotting, AutoCAD instructed the plotter to plot all blue-colored objects with whatever pen was in pen holder #3. Pens could be assigned linetypes, colors, and widths, depending on the capabilities of the plotter.)



Color-dependent plot style tables have exactly 256 plot styles, one for each ACI color (AutoCAD Index Color). AutoCAD cannot accurately plot true colors (16.7 million) and color books (DIC, Pantone, and RAL) with these style tables; if you must have true colors, however, use the **Use Entity Color** option to plot with the ACI color nearest to the true color



Note If you are not sure whether a drawing uses color-dependent or named plot styles, look at the Plot Style list in the layers palette. If its style names are grayed out, color-dependent styles are in effect; if not, named plot styles are in effect.

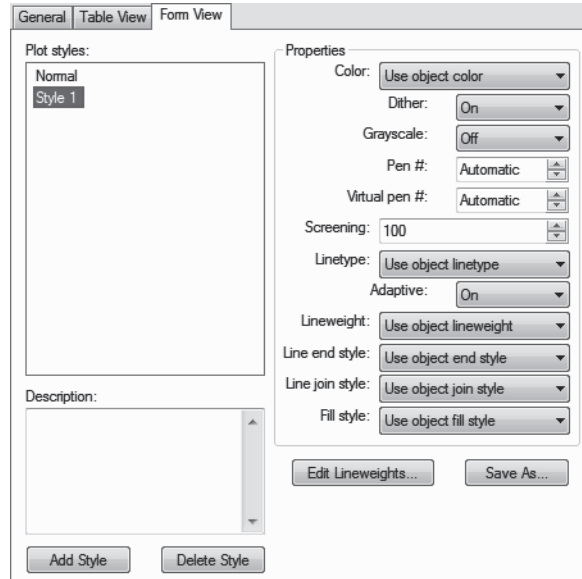


Left: Grayed-out names indicating color-based plot styles are in effect.
Right: Indicating named plot styles are available.

NAMED PLOT STYLE TABLES (.STB FILES)

Named plot style tables (.*stb* files) contain named plot styles assigned to layers and objects. This is much more flexible than color-dependent styles, because it is not limited to controlling objects by their color. Instead, every layer and object can have its own plot style.

You can assign a different named STB to each layout in the drawing.



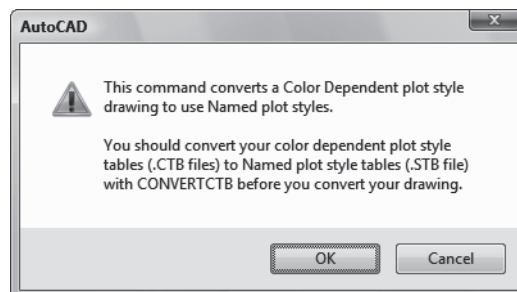
CONVERTCTB AND CONVERTPSTYLES

You can switch drawings between color-dependent and named plot style tables with the **CONVERTPSTYLES** command (short for “convert plot styles”).

From Color-dependent to Named

When a drawing changes from color-dependent to named plot styles, AutoCAD removes CTB color-dependent plot style tables from layouts, and replaces them with named ones. Color-dependent plot style tables should first be converted using these steps:

- Step 1: Use the **CONVERTCTB** command to convert color-dependent plot style tables to named tables. Select a .*ctb* file from the dialog box, and then specify the name for the .*stb* file. The 256 converted plot styles are given generic names: *Style1*, *Style2*, and so on.
- Step 2: Use the **STYLESMANAGER** command to rename the generic plot styles. Styles should be renamed *before* being attached to layouts.
- Step 3: Use the **CONVERTPSTYLES** command to switch the drawing to named plot styles. The command has no options, but displays a warning:



Click **OK**, and a moment later AutoCAD confirms the conversion:

Drawing converted from Named plot style mode to Color Dependent mode.

From Named to Color-dependent

When a drawing changes from named plot styles to color-dependent, AutoCAD erases any STB plot style names assigned to layers and objects in the drawing. Using the CONVERTPSTYLES command results in a message similar to that shown above, along with the confirmation message:

Drawing converted from Color Dependent plot style mode to Named mode.

Although it *appears* that the UNDO command reverses the conversion, named plot styles are labeled “missing” and are **not** returned.

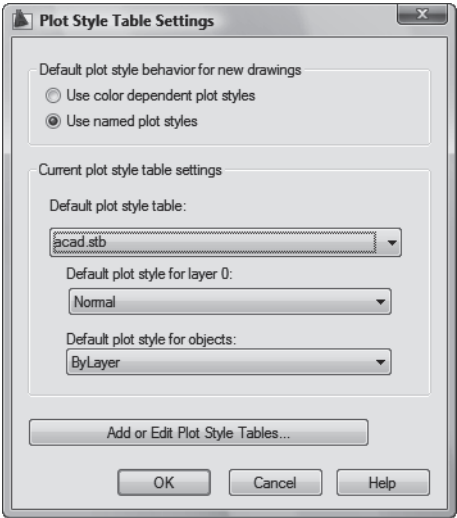
Setting Default Plot Styles

In almost all cases, your drawings should use named plot styles, because they have more benefits than color-based ones. This comparison lists the benefits:

Color-based Tables	Named Tables
256 plot styles	Only as many plot styles as your drawings need. <i>Benefit:</i> Fewer plot styles to deal with.
Generic style names, such as “Color1” and “Color2”	Descriptive style names, such as “50% Screening.” <i>Benefit:</i> Easier to understand style names.
Object colors control for plots	Object color independent of plotting. <i>Benefit:</i> More versatile use of colors in drawings.
Single plot style for all layouts	Every layout having its own plot style. <i>Benefit:</i> Drawing can use layouts for different plotters.

Despite the drawbacks to color-based plot styles, they remain the default setting for new drawings. To change the default plot style table for new drawings through a template file, follow these steps:

- 1. Enter the **OPTIONS** command.
- 2. Choose the **Plot and Publish** tab.
- 3. Click **Plot Style Table Settings**. Notice the Plot Style Table Settings dialog box.



- 4. Select **Use Named Plot Styles**, and then choose a default plot style table (.stb file).
- 5. Click **OK** twice to exit the dialog boxes, and then save the drawing as a template file with the **SAVEAS** command.

Assigning Plot Styles

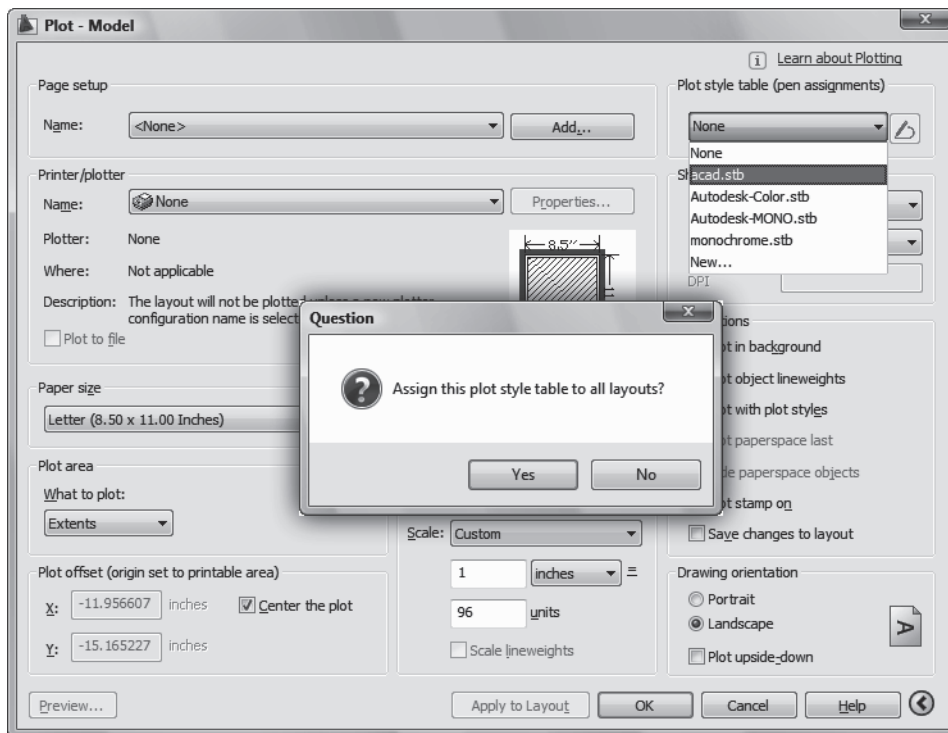
Named plot styles can be assigned to entire layouts, layers, and even individual objects.

Like colors and linetypes, plot styles can be *ByLayer* and *ByBlock*. *ByLayer* means objects take on the plot style assigned to the layer; change the plot style, and all objects on the layer change. *ByBlock* means objects in a block take on the plot style assigned to the block. In addition, a plot style can be *Default*, which means objects take on the plot style assigned to the layout.

Layouts

Assigning a plot style to all layouts takes these steps:

1. Start the **PLOT** command, and then click the **More** button to expand the dialog box.
2. In the Plot Style Table droplist, choose an *.stb* file (plot style name).



3. Notice that AutoCAD asks whether to apply the table to all layouts. You can answer yes or no:
 - **Yes:** the same plot style table is assigned to the model tab and all layouts in the drawing.
 - **No:** this plot style table is assigned only to the model tab.
4. Click **OK** to exit the dialog box.

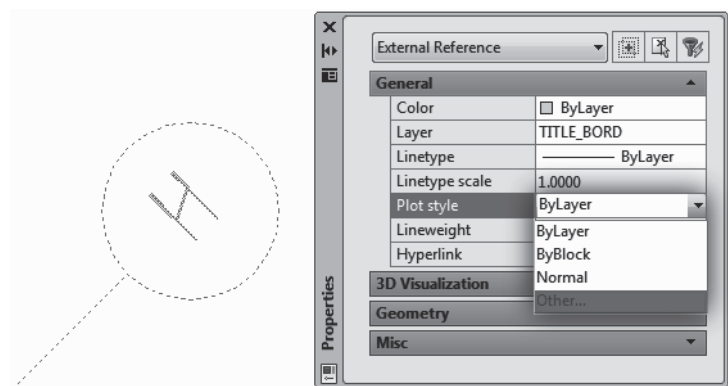
If you want different a plot style table for each layout, then you have to take some more steps:

5. Select a layout tab. If AutoCAD does not automatically display the Page Setup Manager again, right-click the layout tab, and select **Page Setup Manager**.
6. Notice that the Plot style table is "None." Select a table from the Name droplist, and then turn on the **Display plot styles** option.
7. Click **OK** to exit the dialog box.

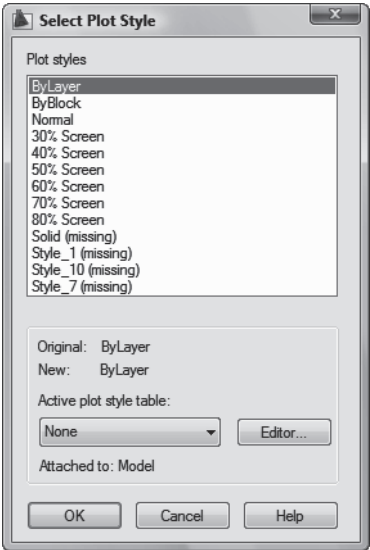
Repeat for each layout tab.

Objects

To assign a plot style to one or more objects, select them in the drawing, and then select the named plot style from the Plot Style Control droplist on the Properties palette.

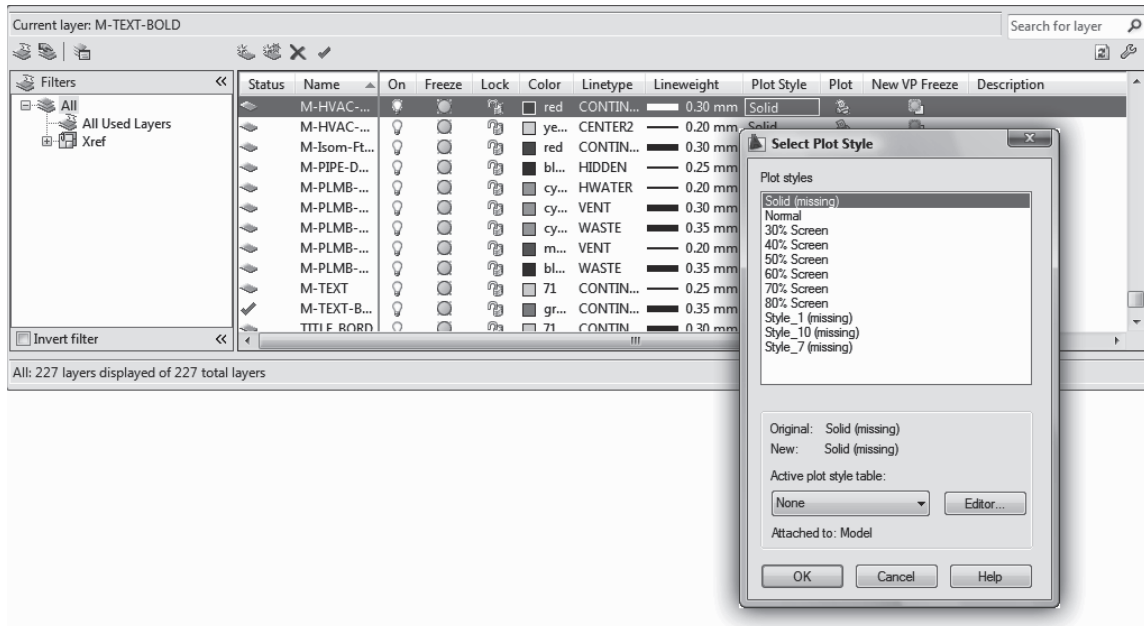


Click **Other** to see a list of all plot styles in the table.



Layers

To assign a plot style to layers, start the **LAYER** command.



In the palette, select one or more layers, and then select a named plot style from the **Plot Style** column. Repeat for other layers and plot styles

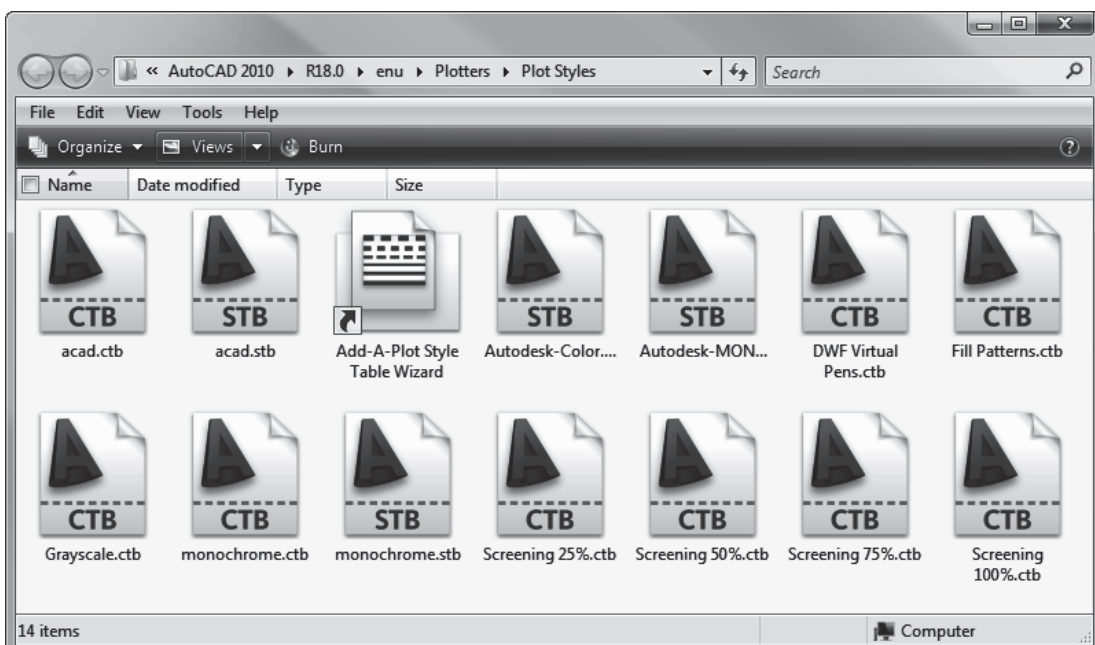
Click **OK** to exit the dialog box. All objects assigned to that layer now take on the same plot style.

TUTORIAL: CREATING NEW PLOT STYLES

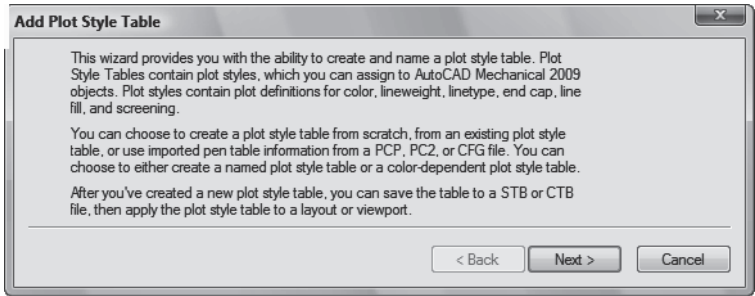
- I. To create new plot styles, start the **STYLESMANAGER** command:
 - At the 'Command:' prompt, enter the **stylesmanager** command:

Command: **stylesmanager** (Press ENTER.)

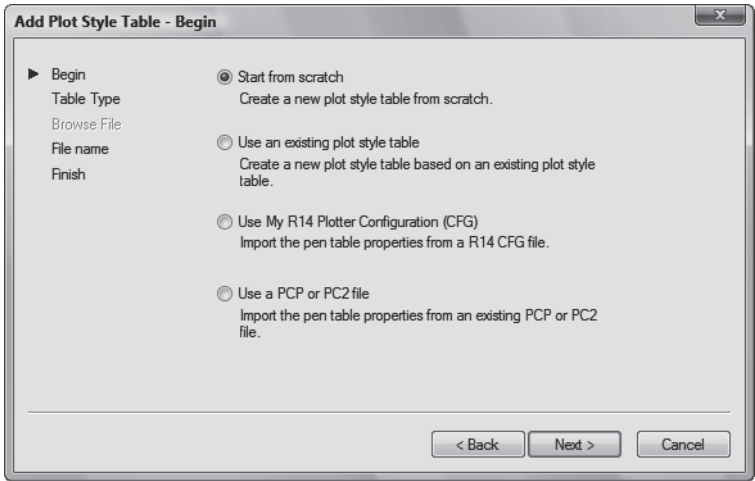
In the window, double-click **Add-A-Plot Style Table Wizard**.



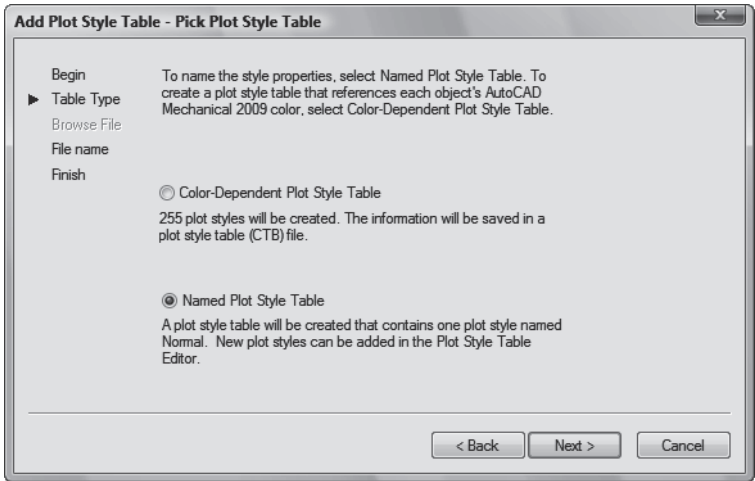
In all cases, AutoCAD runs the Add Plot Style Table “wizard,” which leads you through the steps for creating a new plot style table.



2. Click **Next**.
3. In the Begin dialog box, AutoCAD asks for the kind of plot style table to create. Select **Start from Scratch**, and then click **Next**.



4. In the Table Type dialog box, select **Named Plot Style Table**.



Click **Next**.

5. In the File Name dialog box, enter a descriptive name for the style table.



Click **Next**.

6. In the Finish dialog box, click **Finish**.



AutoCAD adds the .stb style table file to its collection stored in the \plot styles folder.

TUTORIAL: EDITING PLOT STYLES

1. To edit plot styles, start the **stylesmanager** command:
 - At the 'Command:' prompt, enter the **stylesmanager** command.

Command: **stylesmanager** (Press ENTER.)
2. In both cases, AutoCAD requests that Windows display the Plot Styles window – the same one you saw before. In this case, however, you double-click a .ctb or .stb file name:
 - **CTB**: color-based plot style table files.
 - **STB**: named plot style table files.

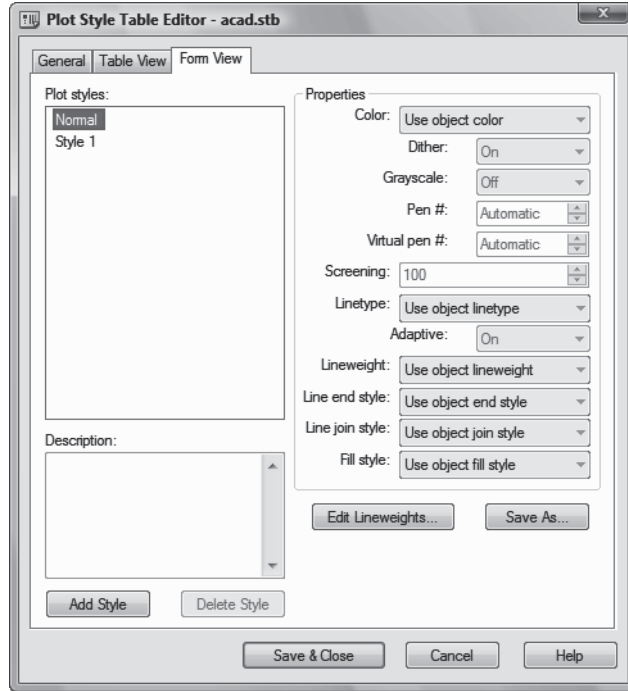
Notice that AutoCAD opens the Plot Style Table Editor.

3. Click the **Form View** tab, which lists plot style names on the left, and the associated plot style properties on the right.

4. Under Plot styles, select a plot style name — other than Normal. Notice the plot style properties at the right.
Edit the properties, as required.



Note The first plot style in every named plot style table is called “NORMAL.” It lists the object's default properties, so that you can see its properties with no plot style applied. The NORMAL style cannot be edited or removed.

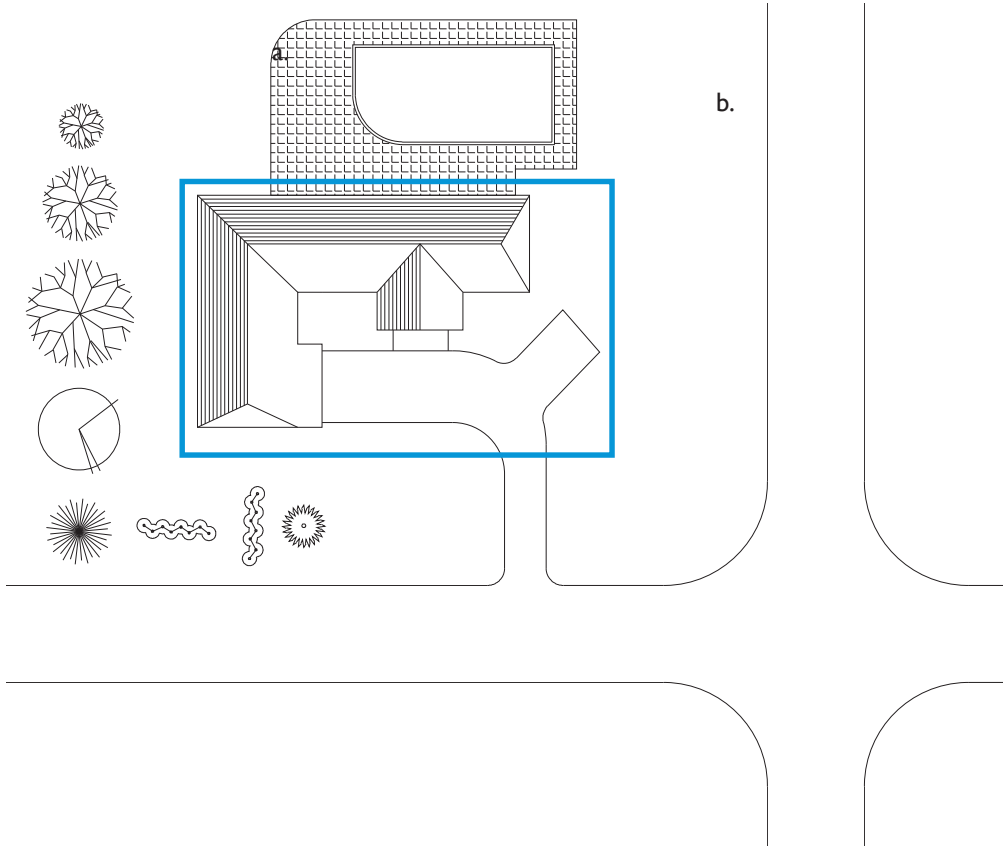


5. To add a plot style, click **Add Style**.
Name the style, and then edit its properties. (You cannot add plot styles to, or delete them from, color-based tables.)
6. To delete a style, select it, and then click **Delete Style**.
7. To save the plot style table, click **Save As**, and then name the table file.
8. When done, click **Save and Close**.

Plot Style Property	Comment
Color	Specifies plotted color of objects; plot color overrides object color. <i>Default: Use Object Color.</i>
Dither	Toggles dithering to approximate colors with dot patterns. <i>Default: Off for .stb files; On for .ctb files.</i>
Grayscale	Converts object colors to grayscale during printing. <i>Default: Off</i>
Pen #	Specifies pen #; ranges from 0 to 32 (available for pen plotters only). <i>Default: Automatic (pen #0).</i>
Virtual Pen #	Simulates pen plotters for non-pen plotters; ranges from 0 to 255. <i>Default: Automatic (pen #0).</i>
Screening	Specifies color intensity; ranges from 0 (white) to 100. Selecting a value other than 100 turns on Dithering. <i>Default: 100.</i>
Linetype	Overrides object linetype with selected linetype. <i>Default: Use Object Linetype.</i>
Adaptive	Adjusts scale of the linetype to complete the pattern. Turn off if linetype scale is crucial. <i>Default: On.</i>
Lineweight	Overrides object lineweight with selected lineweight. <i>Default: Use Object Lineweight.</i>
Line End Style	Assigns a style to the end of object lines: Butt, Square, Round, and Diamond. <i>Default: Use Object End Style.</i>
Line Join Style	Assigns a style to the intersection of object lines: Miter, Bevel, Round, and Diamond. <i>Default: Use Object Join Style.</i>
Fill Style	Overrides object fill style with the following: Solid, Crosshatch, Diamonds, Horizontal Bars, Slant Left, Slant Right, Square Dots, Checkerboard, and Vertical Bar. <i>Default: Use Object Fill Style.</i>

EXERCISES

1. From the Companion CD, open the *edit4.dwg* file, a landscape drawing.



- a. Start the **PLOT** command, and assign a printer, if necessary.
 - b. Plot the drawing at a scale factor of “Scaled to Fit.”
Does the drawing fit the page?
2. Use a ruler to measure the house’s longest side. Assume the length represents 50’.
At what scale was the house plotted?
 3. Plot the drawing again, using both landscape or portrait orientation.
 4. Using the same drawing, set the plot scale factor to 1:1.
Preview the drawing.
Is the drawing larger or smaller?
 5. Plot the drawing again, but this time use the **Window** option, and window the area shown above by the blue rectangle.
 6. Use the **PLOTSTAMP** command to attach a stamp to the plot.
 7. Use the **PUBLISH** command to create a drawing set of the three drawings listed below:
15_43.dwg
15_44.dwg
15_45.dwg

8. Create a plotter configuration for the printer attached to your computer.
Name the configuration "UsingAutoCAD."
9. Create a plotter configuration for outputting compressed TIFF files with the **PLOT** command.
Test the configuration by plotting a drawing, and then viewing the .tif file with AutoCAD's **REPLAY** command.
10. Calibrate the printer attached to your computer.
How inaccurate was it before calibration?
11. Create a named plot style table with the following plot styles:
 - 50% screening.
 - Checkerboard fill.
 - Diamond join style.
 - 0.02 lineweight.Attach the plot style table to a drawing, and then plot it.

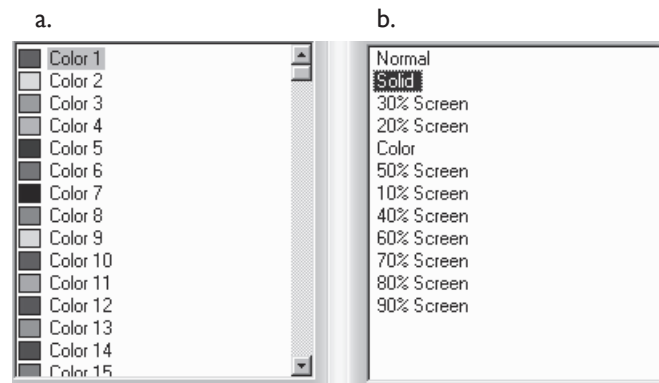
CHAPTER REVIEW

1. What is the purpose of plotting?
2. What part of the drawing is plotted when plotted to the extents of the drawing?
3. Describe how to plot only a portion of drawings.
4. How do you rotate the plot on the paper?
5. Why would you plot drawings to file?
6. Can you plot a drawing without first assigning a plotter or printer?
7. Explain the benefit to using the **PREVIEW** command.
8. Briefly describe the purpose of *device drivers*.
9. Does AutoCAD plot objects on frozen layers?
10. What kinds of plotters and printers does AutoCAD work with?
11. How large is a drawing plotted when the scale factor is
 - a. Scaled to Fit?
 - b. 1:1?
 - c. 1/128" = 1?
12. Name an advantage to using full preview.
To using partial preview.
13. What is the benefit of background plotting?
14. When might you want to offset the drawing to the paper?
What is the drawback of offsetting?
15. Can AutoCAD create rendered plots?
Can 2D drawings be rendered?
16. How do you instruct AutoCAD to plot with hidden lines removed in model space?
In a layout?
17. Describe how plot stamps are useful.
18. When a plot stamp indicates the scale factor is 1:0.97531864, what might this indicate?
19. What are *drawing sets*?
When might you want to use them?
20. Must all layouts in a drawing set come from the same drawing?
21. Explain the purpose of the **PUBLISH** command.
Of the **PLOTTERMANAGER** command.
22. Can AutoCAD plot to printers located on networks?
Connected via USB ports?
23. Explain the meaning of the parts of the printer icon:



- a.
- b.
24. What is the purpose of the *.pc3* file?
The *.stb* file?
The *.ctb* file?
25. Can AutoCAD print on both side of the paper?

26. Explain the purpose of the **STYLESMANAGER** command.
27. Of the two lists of plot style names shown below, which are color-based names and which are named styles?



- a.
- b.
28. Describe two advantages of named plot styles over color-based ones:
- a.
- b.
29. Can named plot styles be applied to layers?
Layouts?
Views?
30. Explain the purpose of the **SCALELISTEDIT** command.
31. Describe how to create a PDF file of a drawing.
32. Why can 3D DWF files be created only from model tab?
33. If a plotted drawing does not fit the paper, what can you do to correct the situation?
- a.
- b.
- c.
34. What might be some reasons for plots failing to show lineweights?
- a.
- b.
- c.

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Appendices

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AutoCAD, Computers, and Windows

The Windows operating system is the foundation on which AutoCAD and other programs run. Windows loads automatically when you start the computer. (You might notice operating system messages and version numbers when your computer starts.) AutoCAD 2010 operates on the Windows XP SP2 or XP x64 Edition, and Vista SP1 or Vista 64-bit operating systems manufactured by Microsoft; although not officially supported yet, AutoCAD also runs on Windows 7.

For XP and Vista, you install a separate version of AutoCAD for 32- or 64-bit systems. The 64-bit version handles larger drawing files.

HARDWARE OF AN AUTOCAD SYSTEM

CAD systems run on computers; *peripherals* connect to computers, as discussed later in this chapter. From smallest to largest, the five primary categories of computer are as follows:

Palm computers are the smallest of computers, fitting in the palm of your hand. Typically, data are input by writing with a stylus on the computer's screen; handwriting recognition software translates writing to text and graphics. Palm computers can be used to view and edit CAD drawings, although AutoCAD itself is not available. CAD software running on PalmOS computers is meant for use at job sites; drawings are exported in DXF format to AutoCAD.

Notebook computers are the smallest practical computers for running AutoCAD. They are the size of a notebook (hence the name), making them transportable. Notebook computers usually consist of a case that contains everything, except the printer. Almost all current notebook computers are powerful enough to run AutoCAD.

Netbook computers can run AutoCAD, even though they are not supported by Autodesk. Because of their slower speed and lower graphics resolution, netbooks are suitable for 2D drawings only; 3D graphics overwhelm this class of portable computer.

Desktop computers are most commonly seen on (or under) desks. These computers usually consist of a case that contains slots for additional adapters, such as for high-end video. The monitor, keyboard, mouse, and output devices (such as printers) are all external to the case.

Workstations are larger, faster, and more expensive than desktop computers. They typically contain two or more fast processors, much more memory, and much larger storage space. This class of

computer is meant for running larger and more sophisticated programs than personal computers typically handle. For example, the special effects of Hollywood movies are created on workstations; some specific brands of CAD software run best on workstations.

Mainframe computers are the largest and most powerful of computers. They are meant for processing huge amounts of data, and are used by governments and companies to process data, such as in making weather forecasts and processing credit card transactions. These computers are not used for CAD.

COMPONENTS OF COMPUTERS

Computers are made up of several essential parts. Let’s look at some of them.

System Board

The system board (sometimes call the “motherboard”) is a fiberglass board that holds most of the computer’s chips and expansion slots. The CPU (central processing unit), RAM memory chips, ROM (read-only memory), drive controllers, and other parts are mounted on this board.

In addition, the board contains slots into which expansion boards can be mounted, such as video capture cards and drive adapters. These are the types of expansion slots:

ISA (industry standard architecture) is the oldest type of expansion slot. The newest computers no longer have these slots.

PCI (peripheral connect interface) is the most common slot in today’s computers, found in both PC and Macintosh computers.

AGP (advanced graphics port) is designed for high-speed data transfer between the CPU and graphics boards.

Dedicated Memory slots are designed for adding more memory to desktop and notebook computers. They are designed to access memory extremely quickly.

Memory Slots are external to computers and read memory cards used by digital cameras and other portable devices.

Internal expansion slots are becoming less important as computer designers integrate most functions into the motherboard, including sound, networking, graphics, and drive interfaces. At the same time, high-speed external interfaces allow you to attach external peripherals to computers, such as hard drives, scanners, and a second monitor.

USB, FireWire, and external memory slots have replaced internal expansion slots, making it possible to add peripherals without ever opening the computer case. USB (universal serial bus) has the advantage of being ubiquitous; it can even recharge portable electronic devices. FireWire has the advantage of providing faster data throughput than USB and imposes no load on the computer’s CPU. Memory cards placed in external memory slots can provide ReadyBoost memory to Windows, which helps speed up its operations.

Other external slots include the following:

PC Card or **PCMCIA** (personal computer memory card interface adapter; also known in jest as “people can’t memorize confusing industry acronyms”) is an older interface designed for notebook computers. The adapter card looks like a thick credit card, and might contain a FireWire interface, wireless network interface, or even a tiny disk drive.

ExpressCard is a smaller and faster replacement for the PC Card interface, and is provided in most notebook computers today. The slot comes in two widths, the rectangular ExpressCard/34 (34 mm wide) and the L-shaped ExpressCard/54 (54 mm wide).

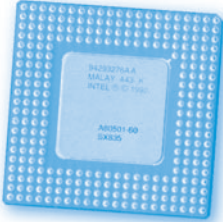
SD and **miniSD** (secure digital) are designed for Palm-size



computers and notebook computers. These cards most often contain memory, but also include wireless networking, GPS (global positioning system), and Bluetooth communications capabilities.

Central Processing Unit

The central processing unit (or CPU) is the “brain” of the computer. This is where the software is processed; and then the CPU sends instructions to the graphics board, printer, plotter, and other peripherals. With a few exceptions, all information passes through the central processing unit. To speed up calculations and access to memory, CPUs are usually packaged with *cache* memory, a very fast form of memory.



Physically, the central processor is a large computer chip mounted on the motherboard. Because of the heat generated by CPUs, it is common to have heat sinks (which look like a series of black fins) and fans mounted on the chips. AMD and Intel manufacture the CPUs commonly used by AutoCAD.

CPUs are also located in other parts of the computer. The computer’s graphics board has its own dedicated CPU for processing graphical images, called the “GPU” (graphics processing unit), and laser printers incorporate CPUs to process print data.

AutoCAD can make use of the graphics board’s GPU for processing visual styles and materials; this is known as “hardware acceleration,” because the GPU is faster than the CPU at these tasks.

Memory

Computer memory can be divided into two categories: ROM (read-only memory) and RAM (random access memory). ROM memory is contained in preprogrammed chips mounted on the motherboard. ROM stores basic sets of commands for the computer, such as the instructions for starting the computer.

Random access memory (RAM) is what most people mean when they refer to “computer memory.” It stores nearly all the information used by the computer while running; it is *temporary*, because all data in RAM are lost when the computer is turned off or when the operating system freezes up.



Software programs (such as AutoCAD) require a certain amount of RAM to run the program. This amount is given in gigabytes. AutoCAD 2009 needs “1GB,” meaning one gigabyte of RAM. A gigabyte is 1,024MB; a megabyte is 1,024KB (kilobytes); a kilobyte is 1,024 bytes.

Hard drive manufacturers often round down these values to make their products appear to have larger capacities, by claiming that there are only 1,000 bytes in a kilobyte, or 1,000,000 bytes in a megabyte.

Disk Drives

Disk drives store large amounts of data, such as application programs, the Windows operating system, drawings, and other documents. Disk drives are either non-removable (*fixed*) or *removable*.

In almost all cases, fixed drives are hard drives. Today’s desktop hard drives hold 240GB and more of data.

Removable drives include diskettes, USB data keys, CDs and DVDs, and even tape drives. The now-rarely-used 3-1/2" diskette holds 1.44MB, a relatively small amount of data; some computers support 2.88MB of data on these diskettes.



USB data keys hold 32MB to 16GB of flash memory, and connect to the USB ports of computers. They are small and expensive relative to other forms of data storage, but have the advantage of being a form of memory that doesn’t lose its data when not attached to a power source.

CD (short for “compact disc, read-only memory”) and DVD (“digital versatile disc”) discs are common for distributing software. They are the same discs that

hold music and movies, but use a different format for coding data. These discs can be read, but not written to. This makes the data secure from accidental erasure and attack from computer viruses. Your copy of AutoCAD probably arrived on a DVD.

Two types of CDs can be written to (called “burning CDs”): CD-R discs can be written to, but not erased, while CD-RW discs can be written to and erased. All CDs allow you to store up to 700MB of data.

As with CDs, there are several types of DVD: “DVD” discs can only be read; “DVD-R” can be written to (recorded); “DVD-RW” can be written to and erased. DVDs hold 4.7GB of data; dual-layer DVDs allow twice as much data to be recorded, while BluRay discs have a capacity of 27GB - 54GB.

DVD drives can read CDs, but CD drives cannot read DVDs. BluRay drives can read DVDs and CDs, but DVD and CD drives cannot read BlueRay discs.

Tape drives were the original backup medium, because they were relatively cheap. They suffer, however, from slow access speed. Today’s tape drives can store 40GB or more data on tapes similar to those used by 8mm digital video cameras.

DISK CARE

All of your work is recorded to disk, so caring for disks is very important. If a disk becomes damaged, you will lose your work! Frequent backing-up (copying files to a second disk drive, tap backup, or recordable DVD-R) and proper handling of disks minimize the possibility of file loss. AutoCAD includes the ability to back up drawings automatically every few minutes to your computer’s hard disk, or safer yet, to the drive on another computer.

Today’s hard drives have self-parking heads that move to an area that does not have stored data; this helps protect disks from damage when moved. Some have sensors that automatically protect the head when a strong gravitational force is felt, such as that of a falling notebook computer.

When the hard drive is on, do not move or tilt the computer or the external drive. This is particularly important with notebook computers. A hard drive can be damaged by shock. If you move the computer or the drive, then move it gently.

CD-ROMs and DVDs are relatively sturdy forms of data storage, but should also be handled with care. Avoid touching either surface of the discs; handle them by their centers and edges. When not in use, keep the discs in cases or protective sleeves; these protect the surfaces from which the laser reads data. Write on the label side with soft-tip pens only; damaging the label with a ballpoint pen damages the data on the disc. Keep discs out of the sun and out of high humidity areas.

There is some controversy over the lifetime of recorded CDs and DVDs. Some claim they last just ten years before the aluminum inside the disc begins to fail; others claim discs can last 100 years or more. In any case, it makes sense to copy important data to new storage formats as they become common.

Write-Protecting Data

Normally, disks can be both read and written to (new data saved on it). But you can *write-protect* some disks to preserve what is already stored on them. This allows the disk to be read by the computer, but not written to. Windows allows you to set *read-only* status for files, folders, and entire drives.

CD-ROMs and DVDs cannot be overwritten, so the data are always safe — except when damaged physically. Some hard drives include utility software that electronically *locks* the drive, preventing misuse.

Some drives can erase data from CD-R and DVD-R





disks, while all DVD-RW and CD-RW disks can have data erased and added. These disks have no write-protect switch, so it is possible to lose data. SD memory cards have a slider switch to prevent data from being overridden, as illustrated by “LOCK” at left.

AUTOCAD 2010

HARDWARE & OS REQUIREMENTS

32-bit AutoCAD 2010

- Windows XP SP2:
 - 1.6GHz CPU with SSE2*;
 - 2.6GHz dual core for 3D modeling
 - 2GB RAM or more
 - 1GB free disk space for installation;
 - DVD-ROM drive for installation
 - 1024x768 16-bit graphics board
 - 1280x1024 32-bit for 3D modeling
 - Internet Explorer 7.0 or higher
- Windows Vista SP1:
 - 3.0GHz CPU with SSE2*
 - 2GB RAM or more
 - 1GB free disk space for installation;
 - DVD-ROM drive for installation
 - 1024x768 16-bit graphics board
 - 1280x1024 32-bit for 3D modeling
 - Internet Explorer 7.0 or higher

64-bit AutoCAD 2010

(64-bit AutoCAD cannot be installed on 32-bit Windows operating systems.)

- XP Professional x64 Edition or Vista 64-bit
 - AMD 64 or Intel EM64T CPU with SSE2*

*) **SSE2** allows AutoCAD to use the CPU's hardware for certain math functions and vector transformations (short for “streaming single-instruction multiple-data extension 2”).

PERIPHERAL HARDWARE

Peripheral hardware consists of add-on devices that perform specific functions. A properly-equipped CAD station consists of the following peripheral devices.

PLOTTERS

Plotters produce “hard” copies of your drawings on paper, vinyl sticker material, cloth, and other media. The two primary types of plotter today are inkjet plotters and laser printers. Some offices still have older pen plotters.

Inkjet Plotters

The most common plotter today is the *inkjet plotter*, which prints by sending an electrical current to the print head. In the print head, electricity heats up ink, causing it to squirt out the end of dozens of nozzles, hitting the paper and soaking in. Typical print heads have 48 nozzles lined up vertically. Print heads in large-format inkjet plotters can have as many as 512 nozzles.

For color plots, there are at least four print heads: one each for cyan (light blue), magenta (pink), yellow, and black. By combining these four colors using a process called *dithering*, inkjet plotters seem to produce 256 or more colors. For higher quality output, specialty inkjet plotters use seven or more colors of ink.

Desktop inkjet printers produce plots of A (8.5" x 11") or B-size (11" x 17"), while floor models plot up to E-size (36" x 48"). Inkjet printers have a typical resolution of 360 dots per inch (*dpi*), but some are capable of 1440 dpi and even 2880 dpi. Higher resolutions are generated by squirting smaller droplets of ink, and placing the droplets closer together. Inkjet plotters can use normal paper, but specially-coated paper results in a cleaner looking print with longer lasting and brighter colors.

When you need paper plots to last many years, make sure your inkjet printer uses pigment inks and archival-quality paper.

Laser Printers

Laser printers create plots using a process similar to photocopying: using a laser beam, the printer “paints” the image of text and drawings on its internal drum. This creates an electrostatic charge on the drum. As the drum rotates, it picks up black toner particles that stick to the electrically-charged portions of the drum. The toner particles are transferred to the paper, which then passes over a hot fuser to melt the toner onto the paper.

The laser-copy process produces sharp, clear prints. Although some laser printers have excellent graphics capabilities, they are usually restricted to a maximum of B-size. Resolution is typically 600 dpi, but some models boast 1200 dpi or higher.

GRAPHICS BOARDS

Graphics boards convert the images generated by AutoCAD into what you see on the monitor. Just about any graphics board made in the last five years works with AutoCAD, but may not support all of its 3D capabilities.

Some of the advanced renderings and visual styles in AutoCAD require advanced capabilities in graphics boards — such as shadow casting and high quality transparency. Autodesk maintains a Web site that lists graphics boards it has tested: www.autodesk.com/autocad-graphicscard. Autodesk ranks each on three levels: Not Supported; Supported, but not Recommended; and Supported and Recommended.

At the time of writing this book, only certain graphics boards from AMD ATI and nVidia are fully compatible with the graphic demands of AutoCAD’s imaging. Boards from other vendors might not be supported.

You can test the graphics board in your computer by running the **3DCONFIG** command, which determines its capabilities for the following:

- Enhanced 2D performance.
- Enhanced 3D performance (geometry acceleration).
- Per-pixel lighting.
- Gooch hardware shader.
- Full shadow display.
- Texture compression.

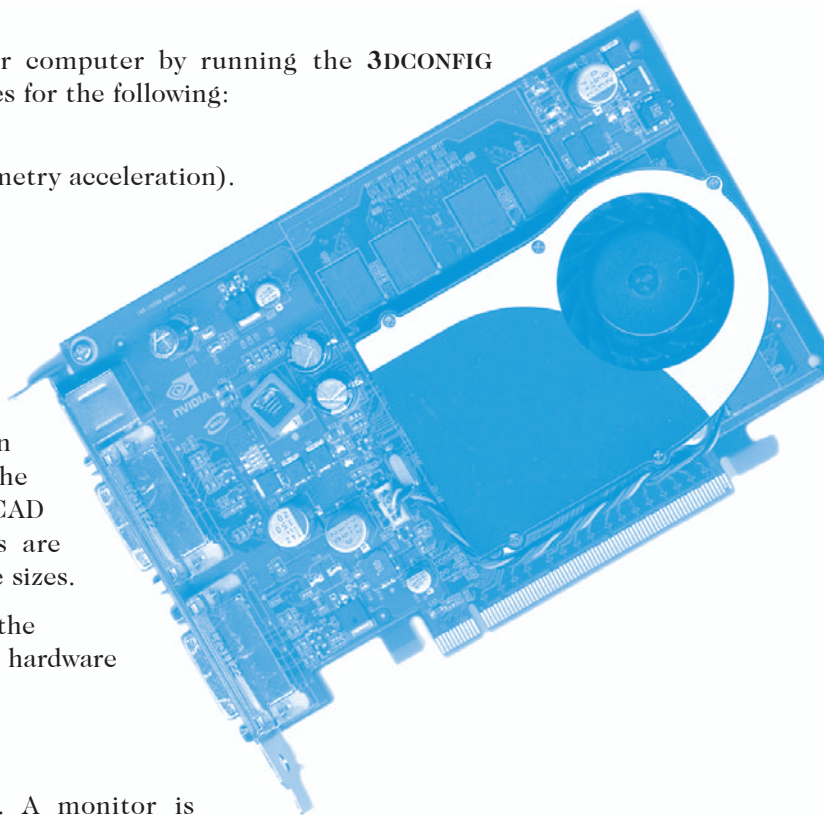
The technical editor observes that AutoCAD tends to be very conservative in rating graphics boards. He has overridden the default values without problems. The author, however, has experienced AutoCAD crashing when the recommended values are exceeded. This may be a function of 3D file sizes.

For truly problematic cases, AutoCAD has the **/nohardware** startup switch that overrides hardware acceleration.

Monitors

Monitors display the work in progress. A monitor is sometimes referred to as a “display device,” “LCD” (liquid crystal display), or “CRT” (cathode ray tube). CAD operators prefer monitors that measure 19 inches (diagonally) or larger, because they can see more of the drawing at a time.

The quality of the image on the monitor is determined by its resolution, which depends on the number of dots (pixels) displayed on the screen. The pixels make up the image. A typical 1280 x 1024-resolution display contains 1,280 pixels horizontally and 1,024 pixels vertically. Many

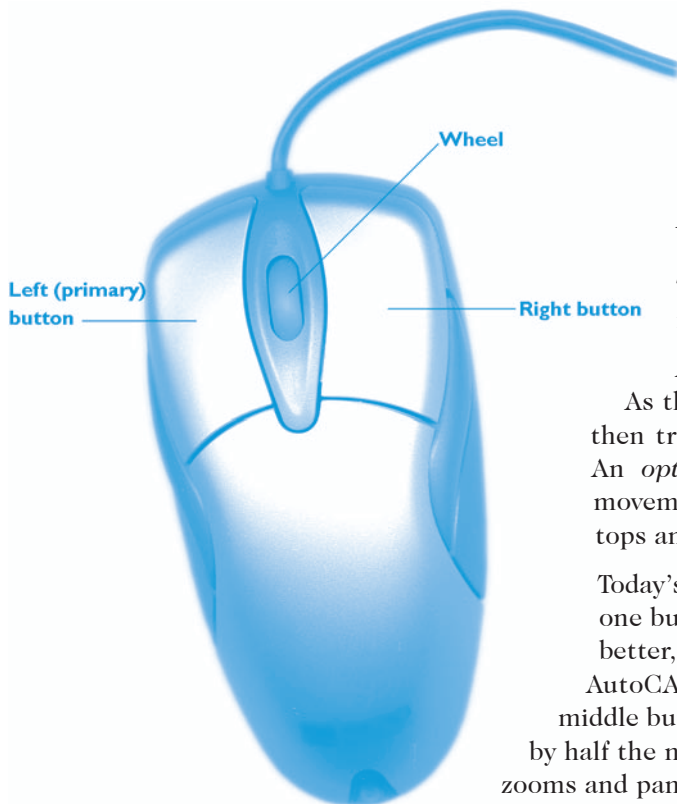


professional CAD users prefer higher resolution displays, such as 1600 x 1200 or higher. The device driver, the operating system, the graphics board, and the monitor work together to support the high resolution.

A few graphics boards support two or even four monitors. The advantage to multi-monitor setups is that one monitor can display the drawing, while the second contains AutoCAD's toolbars, palettes, and other user interface elements. The third and fourth monitors could be used to display other software, or simply spread out the AutoCAD window. The drawback to multiple monitors is the added cost and loss of desk space.

INPUT DEVICES

Just as word processing programs require keyboards to input letters, numbers, and symbols, CAD programs require input devices to make and manipulate drawing elements. In addition to the keyboard, the most common input devices for CAD drawing are the mouse, digitizer, and scanner.



Mouse

The mouse is an input device used for pointing and positioning the cursor. (The name “mouse” comes from its mouse-like appearance; cordless mice dispense with the tail, relying on a small transmitter to connect with the computer.) A mouse can be used for command input and screen interaction; it cannot digitize drawings. There are primarily two types of mouse: mechanical and optical.

A *mechanical* mouse has a ball under its housing.

As the ball rolls, the mouse measures its motion, and then transmits the relative movement to the computer. An *optical* mouse uses optical technology to sense movement over almost any surface — except glass table tops and other shiny surfaces.

Today’s mouse usually sports three or more buttons, with one button disguised as a roller wheel. More buttons are better, because they can be used for additional functions;

AutoCAD supports up to 15 buttons. For example, the middle button can be made to issue a double-click, reducing by half the number of clicks. The roller wheel allows real-time zooms and pans in AutoCAD.

Digitizer

The digitizer is an electronic input device that transmits the *absolute* x and y location of the *puck* resting on its sensitized pad. Digitizers have a fine grid of wires sandwiched between glass layers. When the puck is moved across the pad, the absolute location is read and transmitted to the computer.

Digitizers can be used in two ways: as *pointing devices* to move the cursor around the screen (like a mouse), or as *tracing devices* to copy drawings into the computer at scale and in proper proportion. In “tablet” mode, the digitizing pad is calibrated to the actual absolute coordinates of the drawing. When used as a pointing device, the tablet is not calibrated.

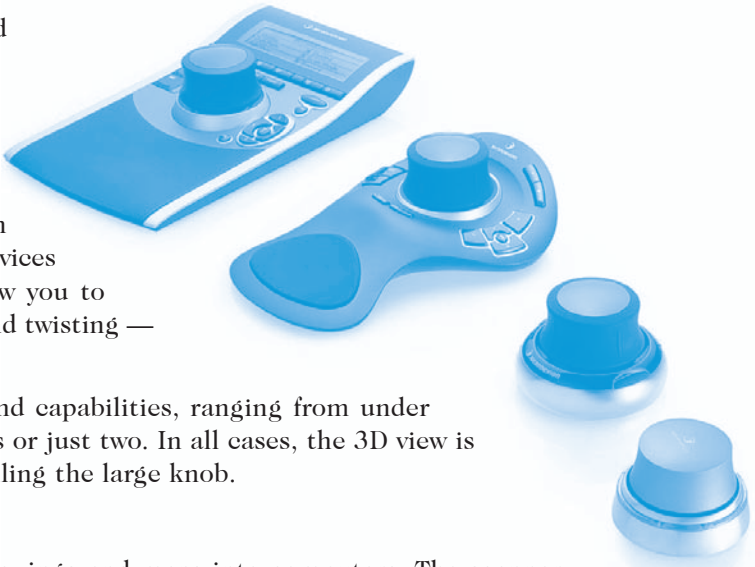
Digitizers can be used with a stylus (similar in appearance to a pencil) or with a multi-button puck. Digitizers come in several sizes, ranging from a few square inches, to very large. Small pads can be used to digitize large drawings: moving the drawing on the pad and then recalibrating. This becomes annoying when you frequently work with large-scale drawings. In that case, the expense of a large-format digitizer (36" x 24" or larger) may be justified.

Digitizers are now very rare, used primarily for tracing existing maps.

3D Mice

One of the problems of working with 3D drawings is that mice and tablets do not easily support view rotation in three dimensions. Specialized input devices from 3dconnexion and Spaceball allow you to change the view by tilting, rotating, and twisting — in addition to zooming and panning.

3D mice come in a range of prices and capabilities, ranging from under \$100 to over \$300, with many buttons or just two. In all cases, the 3D view is controlled by twisting, pushing, or pulling the large knob.



Scanners

Scanners are used to “read” paper drawings and maps into computers. The scanner works by shining a bright light at the paper, while a head moves across the paper, measuring the amount of reflected light. (The head is made of many CCDs, the same charge couple devices used by digital cameras). Bright areas are the paper, while dark areas are lines.

The scanner sends data to the computer in raster format, where they can be imported by AutoCAD with the IMAGEATTACH command, or else converted to vector files by specialized software. Scanners are available in a variety of sizes, ranging from very small units that read business cards to E-size scanners that read large engineering drawings.

DRIVES AND FILES

Windows names the computer’s disk drives with letters of the alphabet, followed by the colon symbol (:), such as A: and C:. A: and B: are reserved for two floppy disk drives — even when your computer has no floppy drives. C: almost always designates the first hard drive, while D: and subsequent letters are for all other drives, including additional hard drives, network drives, CD and DVD drives, and removable drives.

Computers access *network drives* over networks. Computers must be connected with a network cable or a wireless network connection to share drawings and other files. The names of network drives are usually prefixed by a double backslash and their computer’s name, such as \\Kat\C. The networked version of AutoCAD allows a single copy to be used by two or more operators, depending on the terms of the network license.

Removable drives are any disk drives from which the medium can be removed. When reading or writing using removable disks, you must be careful that the correct disk is inserted. (Removable drives are sometimes called “backup drives.”)

The position of the drives in the computer can differ according to the cabinet design. The figure on the next page illustrates typical locations.

FILE NAMES, EXTENSIONS, AND PATHS

Files have names of up to 255 letters and characters. At the end of the file name you often find a dot (.) and a three-letter code, called the *file extension*. This denotes the type of file. For example, AutoCAD drawing files have the .dwg extension, while AutoCAD template files use the .dwt extension.

Many programs, such as AutoCAD, add file extensions to file names automatically, while others require you to specify the extension. The display of file extensions is normally turned off by Microsoft. Because it is useful to identify files by their extension, I recommend that you turn on the display:

In Windows Explorer, select **View | Folder Options**. In the dialog box, click the **View** tab, and then uncheck the **Hide extensions for known file types** option. Similarly, turn on **Hidden Files** to see AutoCAD support files, such as templates.

The full file name includes the *path*. The path specifies the names of the drive and folders in which the file is stored. For example, an AutoCAD drawing named *widget* stored on the C: drive in the *\autocad\drawings* folders is fully named by its path as:

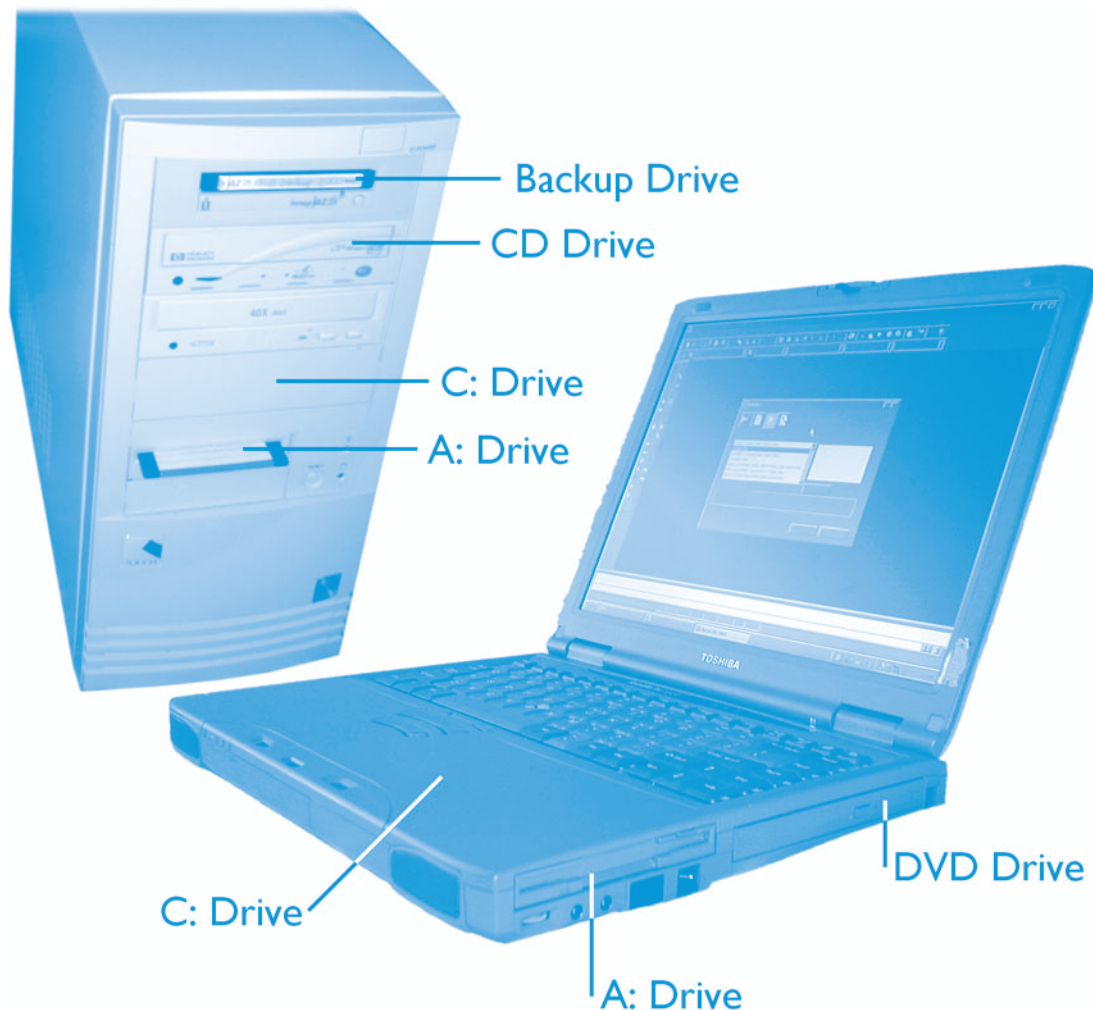
`c:\autocad\drawings\widget.dwg`

Sometimes you might see file names that are just eight characters long and perhaps end in *~1*. Older operating systems, such as Windows 3.1 and DOS, limited file names to eight characters. When a file name of nine or more characters is copied to a computer running one of these older operating systems, the file name is *truncated* (chopped off) to six characters, and *~1* is added. For example, *houseplan.dwg* becomes *housep~1.dwg*. Similar names are truncated to *~2*, and so on.

Wild-Card Characters

In all file dialog boxes, as well as in some Windows functions and certain AutoCAD commands, you can use *wildcard characters* to specify groups of files. The two most common wildcard characters are the question mark (?) and the asterisk (*).

The **question mark (?)** fills in for *any single* character, even if the character is nothing. By using



The location of disk drives in typical desktop and notebook computers.

one or more question marks, you set an upper limit to the number of characters.

When you enter:

car??

you refer to variations on “car...”, such as:

car cars carts card2

But not “cardindex,” because it has more than five characters.

The **asterisk** (*****) represents *any number of* characters. For example, when you enter:

*.dwg

you are referring to all files with the “...*dwg*” file extension, such as

floorplan.dwg my drawing 1.dwg 8th floor.dwg

Alternately, when you type

floorplan.*

you refer to all files named “Floorplan...” regardless of their file extension, as well as to those with no extension:

floorplan.dwg floorplan.bak floorplan.dwf floorplan.

DISPLAYING FILES

Windows provides several ways to display and manipulate files on your computer.

Windows Explorer

Windows Explorer is the “official” method to view files in Windows. Not to be confused with Internet Explorer, Windows Explorer lets you view files in all folders on all drives, including on computers networked to yours.

File-Related Dialog Boxes

Almost all of AutoCAD’s file-related dialog boxes — such as those opened by the **OPEN**, **SAVE AS**, and **IMPORT** commands — let you view and manipulate files. This is a handy shortcut that obviates the need to switch to Explorer.

DOS Session

The oldest method is to start a DOS session, and then type in commands — such as the **DIR**, **TREE**, **COPY**, **MOVE**, and **DELETE** commands— together with parameters, such as **/s**, ***.***, and **a:**.

This approach appeals to power users, because Windows Explorer cannot perform some file-related tasks, such as renaming all files in a folder, or copying files to devices, such as **lpt:**.

WINDOWS EXPLORER

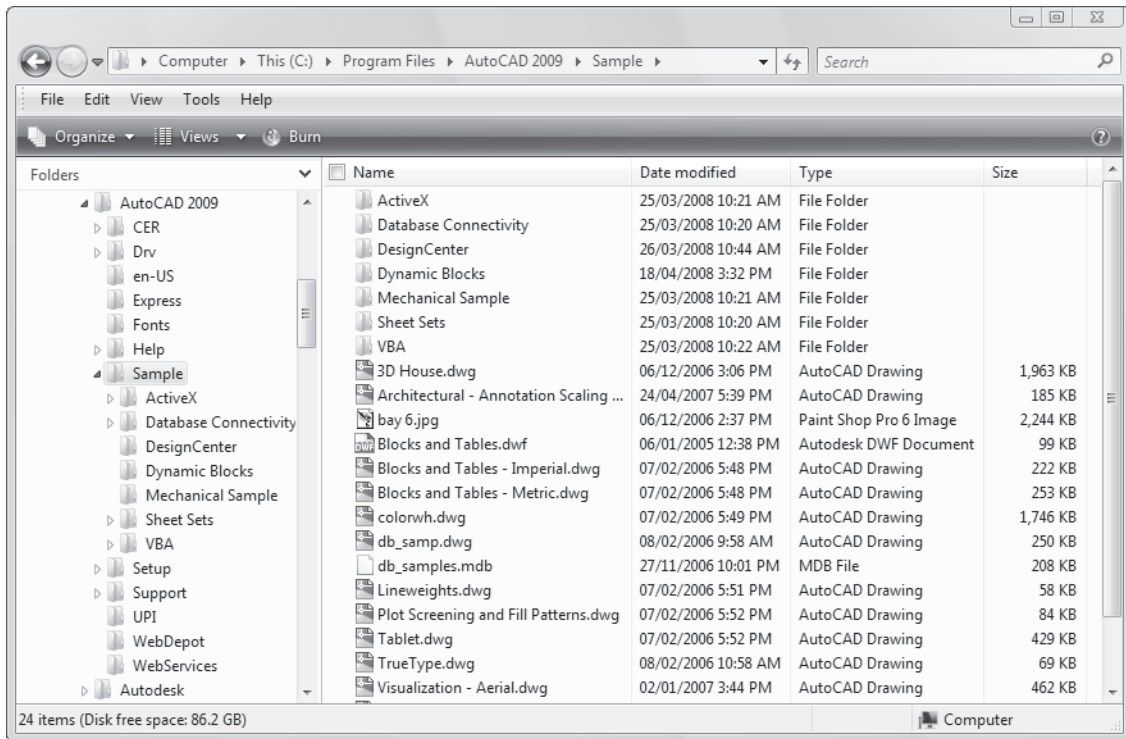
Windows Explorer lets you view and manipulate files on your computer’s drives and all networked drives that you have permission to access. You can change the way Explorer looks, but the most useful configuration has two panes, as illustrated by the figure below.

Viewing Files

In Windows Explorer, the left *pane* displays the *tree* of drives (including networked drives) and folders (replacement name for subdirectories). This is called the “Folder view.” To see more of the tree, use the scroll bar.

The **+** (plus) sign beside a folder means it has subfolders. Click **+** to open the folder; the **+** changes to a **–** (minus) sign. Click **–** to close the folder. Vista uses triangles to open and close folders.

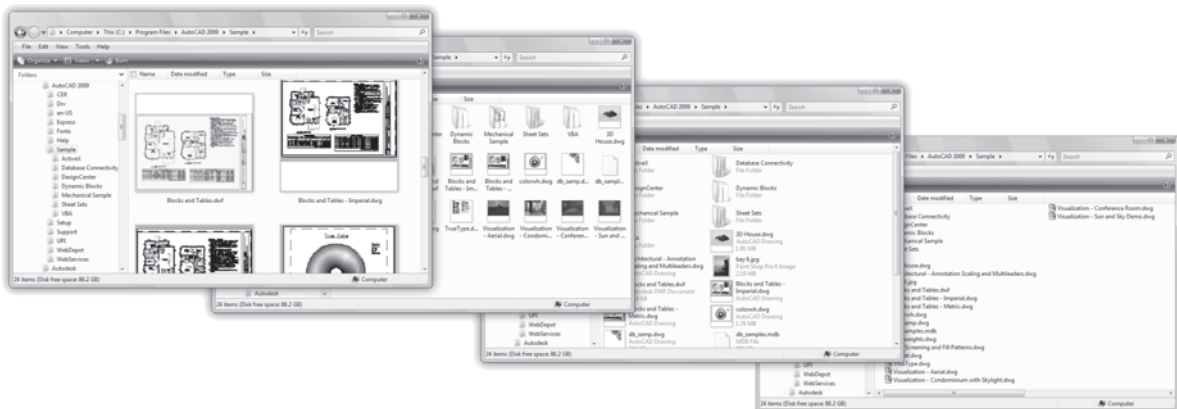
In the right pane, Windows Explorer displays the files and subfolders contained in the folder currently selected in the left window pane. This is called the “Contents view.”



Windows Explorer showing two panes and file details.

You change the sorting order of the **Name**, **Size**, **Type**, and **Modified** lists by clicking on the sort bar. (To use the sort bar, set the **View** to display file **Details**.) Click a second time and Explorer displays the list in reverse order, which is useful for displaying files sorted alphabetically or by size, beginning with either largest or smallest.

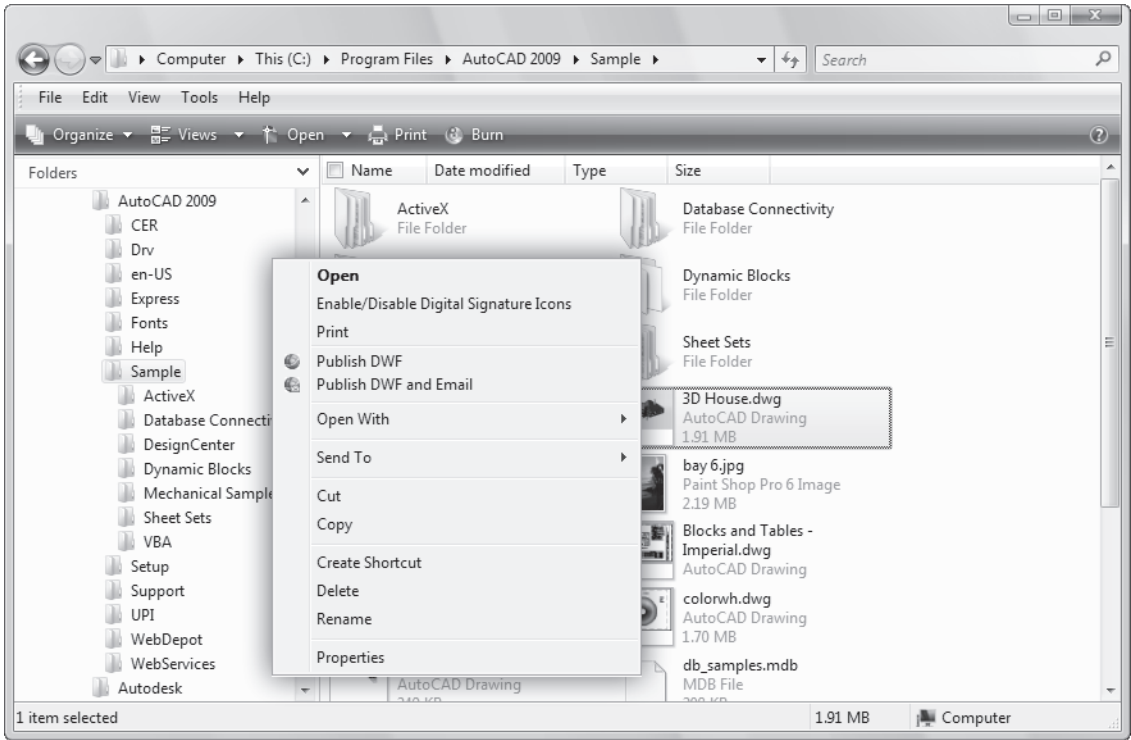
Windows Explorer also displays files and folders in different views, each giving you more detail. The **View** item on the menu bar lets you switch between icons, lists, details, and thumbnails, as illustrated by the figure below. Thumbnail view is useful, because it previews AutoCAD drawing files.



Manipulating Files

By right-clicking on the name of any file or folder, you can change its name and perform other tasks. (Alternatively, use the **File** and **Edit** items on the menu bar.) When you right-click, Windows Explorer displays a shortcut menu with the following options, which differ depending on whether you right-click a folder or file:

Open opens the file with the associated program. For example, a *.lsp* (AutoLISP source code) file is opened by Notepad. An *.exe* file, such as *acad.exe*, starts to run.



Enable/Disable Digital Signature Icons toggles the display of digital signature icons on AutoCAD drawing files. (The icon indicates that the file is protected by a digital signature.)

Print opens the drawing in AutoCAD, and then starts the **PLOT** command for printing the drawing.

Publish DWF opens the drawing in AutoCAD, and then starts the **3DDWF** command for conversion to 3D DWF or DWFx format.

Publish DWF and Email opens the drawing in AutoCAD, and then starts the **3DDWF** command; after the drawing is saved in DWF format, AutoCAD launches your computer's default email software and adds the DWF file as an attachment.

Open With allows you to choose another application in which to open the drawing file.

Send To copies the drawing file to another location, such as a Bluetooth device, a ZIP file, or a PalmPilot.

Cut and **Copy** copy the file's location to the Clipboard; use the **Paste** command to place the file in another folder. Cut moves the file; Copy copies it.

Create Shortcut places *shortcuts* as icons on the Windows desktop. Shortcuts are small files that point to the original file's location. They are excellent for accessing files without wandering through the myriad of Windows' folders to locate them each time.

Delete erases files. Windows first asks, "Are you sure?" Deleted files and folders are stored in the Recycle Bin for a limited time, from which they can be recovered — usually. The files are gone for good when of a very large size, or when deleted from a drive connected over the network.

Rename changes the names of files. As an alternative, click twice (slowly, do not double-click) on the name and Explorer lets you change the name directly.

Properties sets the attributes for the file, including archive, read-only, hidden, and system.

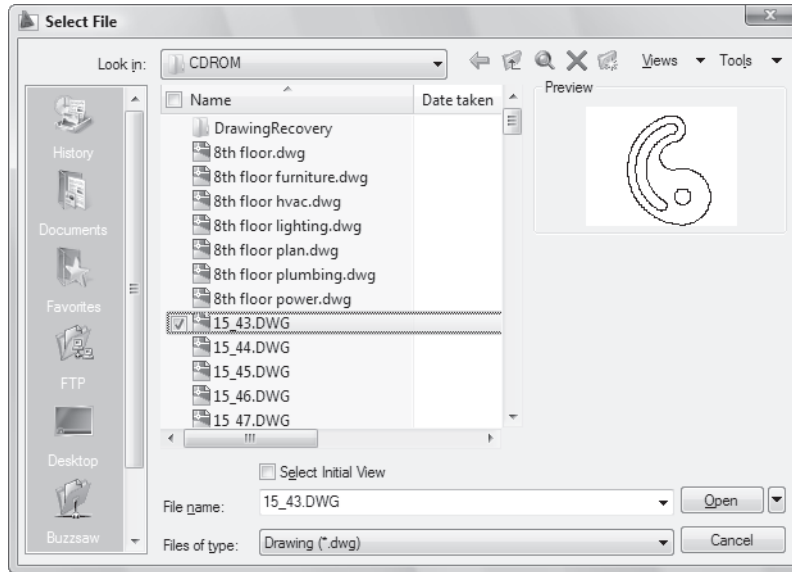
If you drag the file on top of an application, the application will attempt to open the file. For example, dragging a *.dwg* file onto the *acad.exe* icon causes AutoCAD to launch with that drawing.

AUTOCAD FILE DIALOG BOXES

Windows Explorer is the official way to view and manipulate files in Windows. You can, however, perform some of these functions from within AutoCAD's file-related dialog boxes.

This makes a lot of sense, because you're often saving and opening file from inside AutoCAD. File-related dialog boxes are displayed when you use the **OPEN**, **SAVEAS**, **EXPORT**, and other commands that involve files.

AutoCAD's File dialog box provides the following functions:



Views lists files in different views, such as icons or with details (size, type, and modified date-time). Click the column headers to sort in alphabetical order or by size, date, and so on. Click a second time to sort in reverse order.

Look In selects a different folder by several methods: (1) the **Look in** list box lets you move directly to another folder; (2) single-click the **Up One Level** icon to move up the folder tree one folder at a time; double-click the folder to move down into it, or (3) click the **Back** button to return to the previous folder.

Create a New Folder creates a new folder in the current folder. As an alternative, press **ALT+F**.

Delete erases the selected file(s); you can also press the **DEL** key. If you delete a file by accident, you can go to the Recycle Bin and retrieve the file.

Rename renames files. You can also click twice (slowly) on the file name, and then type a new name.

Cut or Copy move or copy files. You can also drag the file to another folder. Hold down the **CTRL** key to copy the file; hold down the **SHIFT** key to move a file.

Open the File by selecting it, and then clicking **OK**. Alternatively, you can double-click the file name to load it without choosing **OK**.

Places List provides convenient access to frequently used folders. Drag a folder from the File list to the Places list to add more locations.

Right-click a file or folder to bring up the same context-sensitive menu as seen in Windows Explorer.

AUTOCAD FILE EXTENSIONS

AutoCAD uses many types of files, the contents of which are described by the file’s extension. For example, linetypes are stored in files that end with *.lin*. Extensions used by AutoCAD are listed here.

Drawing Files

Extension	Description
.bak	Backup drawing files.
.dwf	Drawing Web format files.
.dwg	AutoCAD drawing files.
.dws	CAD standards files.
.dwt	Drawing template files.
.dxb	AutoCAD binary drawing interchange files.
.dxf	AutoCAD drawing interchange files.
.dwfx	XPS compatible version of DWF.

Support Files

Extension	Description
.\$ac	Temporary files created by AutoCAD.
.aws	AutoCAD workspace files.
.blk	Block template files.
.cfg	Configuration files.
.chm	HTML format help files.
.cui	Customize User Interface files.
.cus	Custom dictionary files.
.det	Dictionary files.
.dst	Sheet set data files.
.dsd	Drawing set description files.
.err	Error log files.
.fde	Field catalog files.
.fmp	Font mapping files.
.hlp	Windows-format help files.
.ies	Illumination distribution data files.
.lin	Linetype definition files.
.lli	Landscape libraries.
.log	Log files created by the LOGFILEON command.
.mli	Rendering material library files.
.mln	Multiline library files.
.mnc	Compiled menu files (obsolete as of AutoCAD 2006).
.mnd	Uncompiled menu files containing macros (obsolete).
.mnl	AutoLISP routines used by AutoCAD menus (obsolete).
.mns	AutoCAD-generated menu source files (obsolete).
.mnu	Menu source files (obsolete as of AutoCAD 2006).
.msg	Message files.
.pat	Hatch pattern definition files.
.pgp	Program parameters files (external commands and aliases).
.scr	Script files.
.shp	Shape and font definition files.
.shx	Compiled shape and Autodesk font files.
.ttf	Microsoft font files.
.xpg	Xml-format tool palette group files.

AutoCAD Program Files

Extension	Description
.arx	ObjectARx (AutoCAD Runtime eXtension) program files.
.dll	Dynamic link library files.
.exe	Executable files, such as AutoCAD itself.

Plotting Support Files

Extension	Description
.ctb	Color-table based plot parameter files.
.pep	Plot configuration parameters files for AutoCAD Release 14.
.pc2	Plot configuration parameters files for AutoCAD 2000.
.pc3	Plot configuration parameters files since AutoCAD 2000i.
.plt	Plot files.
.stb	Style-table based plot parameter files.

Import-Export Files

Extension	Description
.3ds	3D Studio files.
.bmp	Windows raster files (device-independent bitmap).
.cdf	Comma delimited files (created by EATTEXT).
.dgn	MicroStation V8 and V7 design files (created by DGNEXPORT).
.dxe	Data extraction files (created by DATAEXTRACTION).
.dxx	DXF files (created by ATTEXT).
.kml	Google Earth files (keyhole markup language).
.kmx	Compressed DML files (read by GEOGRAPHICLOCATION).
.pex	Raster format file.
.png	Portable Network Graphics raster files.
.sat	ACIS solid object files (short for “Save As Text”).
.slb	Slide library files.
.sld	Slide files.
.stl	Solid object stereo-lithography files (solids modeling).
.tif	Raster format files (Tagged image file format).
.tga	Raster format files (Targa).
.txt	Space delimited files (created by EATTEXT).
.wmf	Windows metafile files.
.xls	Excel spreadsheet files (created by EATTEXT).
.xml	DesignXML format files.

Miscellaneous Files

Extension	Description
.css	Cascading style sheet files.
.html	HTML files.
.htt	HTML template files.
.ini	Initialization files.
.js	JavaScript files.
.map	Map files displayed by GEOGRAPHICLOCATION.
.xls	XML style sheet.
.xml	Extended markup language files.
.xmx	External messages files.

LISP and ObjectARX Programming Files

Extension	Description
.actm	Active macro source code files.
.cpp	ObjectARX source code files.
.dcl	Dialog control language descriptions of dialog box files.
.dvb	Visual Basic for Applications program files.
.def	ObjectARX definition files.
.fas	AutoLISP fast load programs.
.frm	VBA form definition files.
.h	ADS and ObjectARX function definitions files.
.lib	ObjectARX function library files.
.lsp	AutoLISP program files.
.mak	ObjectARX make files.
.rx	Lists of ObjectARX applications that load automatically.
.tlb	ActiveX Automation type library files.
.unt	Unit definition files.
.vlx	Compiled Visual LISP files.

REMOVING TEMPORARY FILES

While it is working, AutoCAD creates “temporary” files. When you exit AutoCAD, these temporary files are erased. If, however, your computer crashes, these files might be left on the hard disk. You can erase temporary files, but only when AutoCAD is not running. Never erase files with these extensions while AutoCAD is still running:

```
. $$$ . $a .ac$ .dwk .dwl .ef$ .sv$ .swr
```

These files can be erased to free up disk space when AutoCAD is not running. Alternatively, the DRAWINGRECOVERY and RECOVERALL commands help you select which of these files can be reused, following a crash.

AutoCAD Commands, Aliases, and Keyboard Shortcuts

AUTOCAD COMMANDS

The following commands are available in AutoCAD. Commands prefixed with - (hyphen) display their prompts at the command line only and not in a dialog box. Shortcut keystrokes are indicated as CTRL+F4.



This icon indicates the 54 commands that are new in AutoCAD 2010.

The Illustrated AutoCAD 2010 Quick Reference by author Ralph Grabowski provides a complete reference guide to all of AutoCAD's commands, and is available through Autodesk Press.

Command	Description
A	
About	Displays an AutoCAD information dialog box that includes version and serial numbers.
AcisIn	Imports ASCII-format ACIS files into the drawing, and then creates 3D solids, 2D regions, or body objects.
AcisOut	Exports AutoCAD 3D solids, 2D regions, or bodies as .sat ASCII-format ACIS files.
AcMrSelect	Prompts you to select objects; after you exit the command, the objects continue to display as selected by showing highlights, grips, and quick properties (undocumented).
 ActBasePoint	Inserts base points in action macros.
 ActManager	Manages action macro files.
ActRecord	Starts recording user actions as a macro.
ActStop	Stops recording the macro, and then displays the Action Macro dialog box, where you fill in the macro name and other details.
-ActStop	Stops recording the macro, and then prompts you at the command line for the macro's name and other details.

Command	Description
ActUserInput	Adds user input to existing macros. You use this command while recording a macro.
ActUserMessage	Adds dialog-box messages to macros being recorded; displays the Insert User Message dialog box. You use this command while recording a macro.
-ActUserMessage	Adds dialog-box messages to macros being recorded; prompts you at the command line. You use this command while recording a macro.
AdcClose	CTRL+2: Closes the DesignCenter palette.
AdCenter	CTRL+2: Opens DesignCenter; manages AutoCAD content.
AdcNavigate	Directs DesignCenter to the file name, folder, or network path you specify.
 Adjust	Changes the fade, contrast, and other parameters of images and underlays; parameters vary according to the object type (a super command).
AecToAcad	Explodes AEC objects and adds them to a new drawing file.
Ai_Publish	Prompts you for an email and then asks to quit (undocumented).
Ai_Send_Feedback	Opens the default Web browser at Autodesk's Contact Us page (undocumented).
Align	Uses three pairs of 3D points to move and rotate (align) 3D objects.
AllPlay	Plays all show motion views in sequence, without prompting. Useful for running animations in front of an audience (undocumented).
AmeConvert	Converts drawings made with AME v2.0 and v2.1 into ACIS solid models.
AniPath	Creates animations of drawings along paths.
AnnoReset	Resets the location of all scale representations of an annotative object to that of the current scale representation.
AnnoUpdate	Updates annotative objects when styles have changed in the drawing.
 Antz	Controls the ViewCube orientation from the command line; displays all SteeringWheels at once (undocumented).
Aperture	Adjusts the size of the target box used with object snap.
AppLoad	Displays a dialog box that lets you list AutoLISP, Visual Basic, and ObjectARX program names for easy loading into AutoCAD.
Arc	Draws arcs by a variety of methods.
Archive	Archives sheet sets in DWG or DWF format.
Area	Computes the area and perimeter of polygonal shapes.
Array	Makes multiple copies of objects.
Arx	Loads and unloads ObjectARX programs. Also displays the names of ObjectARX program command names.
 Attach	Inserts external references, images, and DWF, DWFx, DGN, and PDF underlays in the current drawing. Xrefs can use geographic data if both the drawing and xref specify longitude/latitude (a super command).
 -Attach	Inserts external files through the command line (undocumented).
AttachURL	Attaches hyperlinks to objects and areas.
AttDef	Creates attribute definitions.
AttDisp	Controls whether attributes are displayed.

Command	Description
AttEdit	Edits attributes.
AttExt	Extracts attribute data from drawings, and writes them to files for use with other programs.
AttIpEdit	Launches the in-place attribute editor.
AttReDef	Assigns existing attributes to new blocks, and new attributes to existing blocks.
AttSync	Synchronizes changed attributes with all blocks.
Audit	Diagnoses and corrects errors in drawing files.
 AutoConstrain	Constrains objects within a preset distance of each other automatically.
AutoPublish	Toggles the simultaneous saving of drawings in DWG and DWF formats.

B

Background	Accesses the Background tab of the View dialog box directly.
Base	Specifies the origin for inserting one drawing into another.
BAttMan	Edits all aspects of attributes in a block; short for Block Attribute Manager.
BEdit	Switches to the block editor; makes the following commands available: BAction — adds actions to parameters in dynamic block definitions.  BActionBar — toggles display of action bars. BActionSet — associates actions with selected objects within dynamic blocks. BActionTool — adds actions to dynamic block definitions. BAassociate — associates actions with parameters in dynamic blocks. BAttOrder — controls the order in which attributes are displayed in dynamic blocks. BAuthorPalette — opens the Block Authoring palette in the Block Editor. BAuthorPaletteClose — closes the Block Authoring palette. BClose — closes the Block Editor.  BConstruction — converts objects between geometry and construction geometry.  BCParameter — applies constraint parameters to selected objects, or converts dimensional constraints to parameter constraints; BC is short for Block Constraint. BCycleOrder — changes the cycling order of grips within dynamic block references. BGgripSet — creates, repositions, resets, and deletes grips associated with parameters. BLookupTable — creates and edits lookup tables associated with dynamic blocks. BParameter — adds parameters and grips to dynamic blocks. BSave — saves changes to dynamic blocks. BSaveAs — saves dynamic blocks by another name.  BTable — defines variations of dynamic blocks; you can copy'n paste an Excel spreadsheet into the table.  BTestBlock — displays a window within the Block Editor to test the dynamic block being edited. BvHide — hides objects that have been assigned the state of invisibility. BvShow — shows objects assigned invisibility. BvState — assigns visibility states to objects in dynamic blocks. BwBlockAs — saves the block as a .dwg file on disc (undocumented).
 BESettings	Displays the Block Editor Settings dialog box.

Command	Description
BHatch	Fills an automatically-defined boundary with hatch patterns, solid colors, and gradient fills; previews and adjusts patterns.
Blipmode	Toggles display of marker blips.
Block	Creates symbols from groups of objects.
BlockIcon	Generates preview images for blocks created with AutoCAD Release 14 and earlier.
BmpOut	Exports selected objects from the current viewport to raster <i>.bmp</i> files.
Boundary	Draws closed boundary polylines.
Box	Creates 3D solid boxes and cubes.
Break	Erases parts of objects; breaks objects in two.
BRep	Removes construction history from 3D solid models.
Browser	Launches your computer's default Web browser with the URL you specify.
C	
Cal	Runs a geometry calculator that evaluates integer, real, and vector expressions.
Camera	Sets the camera and target locations.
Chamfer	Trims intersecting lines, connecting them with a chamfer.
Change	Permits modification of an object's characteristics.
CheckStandards	Compares the settings of layers, linetypes, text styles, and dimension styles with those of another drawing.
ChProp	Changes properties (linetype, color, and so on) of objects.
ChSpace	Moves and scales objects from model to paper space.
Circle	Draws circles by a variety of methods.
ClassicImage	Replaces the IMAGE command dialog box (undocumented).
ClassicLayer	Replaces the LAYER command for the modal layer dialog box, the one where you need to click OK before continuing to edit.
ClassicXref	Replaces the XREF command dialog box (undocumented).
CleanScreenOff	CTRL+0 : Turns on toolbars, title bar, and window borders.
CleanScreenOn	CTRL+0 : Maximizes drawing area by turning off toolbars, title bar, and other elements.
 Clip	Crops external references, images, viewports, and DWF, DWFx, DGN, and PDF underlays to a specified boundary; can invert clips for all reference files (a super command).
Close	CTRL+F4 : Closes the current drawing.
CloseAll	Closes all open drawings; keeps AutoCAD open.
Color	Sets new colors for subsequently-drawn objects.
CommandLine	CTRL+9 : Displays the command line palette.
CommandLineHide	CTRL+9 : Hides the command line palette.

Command	Description
Compile	Compiles shapes and <i>.shp</i> and <i>.pfb</i> font files.
Cone	Creates 3D solid cones.
 ConstraintBar	Displays the available geometric constraints on objects; a toolbar-like element.
 ConstraintSettings	Controls the display of geometric constraints on constraint bars through a dialog box.
 +ConstraintSettings	Specifies which tab of the Constraint Settings dialog box to display (undocumented).
Convert	Converts 2D polylines and associative hatches in pre-AutoCAD Release 14 drawings to the “lightweight” format to save on memory and disk space.
ConvertCTB	Converts drawings from plot styles to color-based tables.
ConvertOldLights	Converts light blocks in drawings created by AutoCAD 2006 and older.
ConvertOldMaterials	Converts material definitions in drawings created by AutoCAD 2006 and older.
ConvertPStyles	Converts drawings from color-based tables to plot styles.
ConvToSolid	Converts closed polylines and circles to extruded 3D solids.
ConvToSurface	Converts 2D solids, regions, lines, polylines, arcs, and flat 3D faces to surfaces.
Copy	Makes one or more copies of objects.
CopyBase	CTRL+SHIFT+C: Copies objects with a specified base point.
CopyClip	CTRL+C: Copies selected objects to the Clipboard in several formats.
CopyHist	Copies Text window text to the Clipboard.
CopyLink	Copies all objects in the current viewport to the Clipboard in several formats.
CopyToLayer	Copies selected objects to another layer through a dialog box.
Cui	Opens the Customize User Interface dialog box.
CuiExport	Exports customization settings to <i>.cui</i> files on disk.
CuiImport	Imports customization settings from <i>.cui</i> files.
CuiLoad	Loads <i>.cui</i> files.
CuiUnload	Unloads <i>.cui</i> files.
 CuiXAppend	Appends <i>.bmp</i> , <i>.dib</i> , and <i>.rle</i> image files to <i>.cuix</i> files (undocumented).
 CuiXCreate	Creates new <i>.cuix</i> files, and then prompts you to add <i>.cui</i> and <i>.bmp</i> files (undocumented).
 CuiXList	Lists root parts and images stored in <i>.cuix</i> files (undocumented).
Customize	Customizes tool palette groups.
CutClip	CTRL+X: Cuts selected objects from the drawing to the Clipboard in several formats.
Cylinder	Creates 3D cylinders.

Command	Description
D	
DataExtraction	Extracts attribute data and drawing information to spreadsheets or tables.
DataLink	Links tables with external spreadsheet files.
DataLinkUpdate	Updates linked tables.
DbcClose	CTRL+6: Closes the dbConnect Manager (undocumented).
DbConnect	CTRL+6: Connects objects in drawings to tables in external database files.
DbList	Provides information about all objects in drawings.
DdPType	Specifies the style and size of points.
DdVPoint	Sets 3D viewpoints.
Delay	Creates a delay between operations in a script file.
 DelConstraint	Removes all geometric and dimensional constraints from a selection set of objects.
DetachURL	Removes hyperlinks from objects.
DgnAdjust	Adjusts the contrast of attached DGN files.
DgnAttach	Attaches MicroStation V8 2D DGN files as underlays.
DgnClip	Clips attached DGN files.
DgnExport	Exports the current drawing as a DGN file.
-DgnExport	Command line version of the equivalent DGNEXPORT command (undocumented).
DgnImport	Imports MicroStation V8 2D DGN files as AutoCAD entities.
-DgnImport	Provides command line version of the equivalent DGNIMPORT command (undocumented).
DgnLayers	Toggles the display of layers (levels) in DGN files placed as underlays in AutoCAD.
DgnMapping	Displays the DGN Mapping Setups dialog box for matching layers, linetypes, lineweights, and colors between MicroStation and AutoCAD.
Dist	Computes the distance between two points.
DistantLight	Places distant lights in drawings.
Divide	Divides objects into an equal number of parts, and places specified blocks or point objects at the division points.
Donut	Constructs solid filled circles and doughnuts.
Dragmode	Toggles display of dragged objects.
DrawingRecovery	Opens the Drawing Recovery palette.
DrawingRecoveryHide	Closes the Drawing Recovery palette.
DrawOrder	Changes the order in which objects are displayed: selected objects and images are placed above or below other objects.
DSettings	Specifies drawing settings for snap, grid, polar, and object snap tracking.
DsViewer	Opens the Aerial View palette.

Command	Description
DView	Displays 3D views dynamically.
DwfAdjust	Changes the contrast, fade, and color of DWF files.
DwfAttach	Attaches <i>.dxf</i> files as uneditable underlays with a dialog box.
DwfClip	Clips DWF underlays.
DwfFormat	Toggles the default format — DWF or DWFX.
DwfLayers	Toggles the display of layers of DWF underlays.
DwgProps	Sets and displays the properties of the current drawing.
DxbIn	Imports binary drawing interchange files.

DIMENSIONS

Dim	Specifies semi-automatic dimensioning capabilities.
Dim1	Executes a single AutoCAD Release 12-style dimension command.
DimAligned	Draws linear dimensions aligned to objects.
DimAngular	Draws angular dimensions.
DimArc	Draws arc dimensions along the circumference of arcs.
DimBaseline	Draws linear, angular, and ordinate dimensions that continue from baselines.
DimBreak	Breaks extension and dimension lines where they cross other objects.
DimCenter	Draws center marks on circles and arcs.
 DimConstraint	Applies dimensional constraints to selected objects or points on objects.
DimContinue	Draws linear, angular, and ordinate dimensions that continue from the last dimension.
DimDiameter	Draws diameter dimensions on circles and arcs.
DimDisassociate	Removes associativity from dimensions.
DimEdit	Edits the text and extension lines of associative dimensions.
DimInspect	Adds inspection text to dimensions.
DimJogged	Draws radial dimensions with jogged leaders.
DimJogLine	Jogs linear dimension lines.
DimLinear	Draws linear dimensions.
DimOrdinate	Draws ordinate dimensions in the x and y directions.
DimOverride	Overrides current dimension variables to change the look of selected dimensions.
DimRadius	Draws radial dimensions for circles and arcs.
DimReassociate	Associates dimensions with objects.
DimRegen	Updates associative dimensions.
DimSpace	Spaces two or more linear dimensions evenly.
DimStyle	Creates, names, modifies, and applies named dimension styles.

Command	Description
DimTEdit	Moves and rotates text in dimensions.
E	
EAttEdit	Starts the enhanced attribute editor.
Edge	Changes the visibility of 3D face edges.
EdgeSurf	Draws edge-defined 3D meshes.
EditShot	Prompts for a view name at the command line, and then displays the Shot Properties tab of the New View / Shot Properties dialog box (undocumented).
EditTableCell	Prompts you to select a table, a cell range, and an operation (undocumented).
Elev	Sets current elevation and thickness.
Ellipse	Constructs ellipses and elliptical arcs.
Erase	Removes objects from drawings.
eTransmit	Packages the drawing and related files for transmission by email or courier.
ExAcReload	Reloads xrefs, and reports whether they had changed.
Explode	Breaks down blocks into individual objects; reduces polylines to lines and arcs.
Export	Exports drawings in several file formats.
 -Export	Exports drawings in DWF, DWFx, or PDF format from the command line (undocumented).
 ExportDwf	Exports drawings as DWF files, and allows you to override individual page setups on a sheet-by-sheet basis through a dialog box.
 ExportDwfx	Exports drawings as DWFx files, and allows you to override individual page setups on a sheet-by-sheet basis through a dialog box.
ExportLayout	Copies layouts to model space.
 ExportPdf	Exports drawings as PDF files, and allows you to override individual page setups on a sheet-by-sheet basis through a dialog box.
 ExportSettings	Exports drawings as DWF, DWFx, or PDF files, and allows you to set export options through an integrated dialog box.
Extend	Extends objects to meet boundary objects.
ExternalReferences	Displays the External References palette for attaching drawings, images, and .dwf files.
ExternalReferencesClose	Closes the External References palette; the longest command name in AutoCAD.
Extrude	Extrudes 2D closed objects into 3D solid objects.
F	
Field	Inserts updatable field text in drawings.
Fill	Toggles the display of solid fills.
Fillet	Connects two lines with an arc.
Filter	Creates selection sets of objects based on their properties.
Find	Finds and replaces text.

Command	Description
FlatShot	Creates 2D views of 3D models.
Fog	Invokes the RENDERENVIRONMENT command (obsolete).
FreePoint	Places target-less point lights in drawings (undocumented).
FreeSpot	Places target-less spot lights in drawings.
FreeWeb	Places target-less web lights in drawings.

G

GeographicLocation Specifies the sun position through date, time, city, longitude, and latitude.



GeomConstraint

Applies geometric relationships between objects or points on objects.

GoToURL	Links to hyperlinks
Gradient	Opens the Hatch and Gradient dialog box at the Gradient tab.
GraphSer	F2 : Switches to the drawing window from the Text window.
Grid	F7 and CTRL+G : Displays the grid at a specified spacing.
Group	Creates named selection sets of objects. (CTRL+SHIFT+A toggles group selection style.)

H

Hatch	Opens the Hatch and Gradient dialog box.
HatchEdit	Edits associative hatch patterns.
Helix	Draws 3D helix objects (spirals).
Help	? and F1 : Displays a list of AutoCAD commands with detailed information.
Hide	Removes hidden lines from the currently-displayed view.
HidePalettes	CTRL+SHIFT+H : Hides all open palettes.
HlSettings	Specifies properties for hidden line removal.
HyperLink	CTRL+K : Attaches hyperlinks to objects, or modifies existing hyperlinks.
HyperLinkOptions	Controls the visibility of the hyperlink cursor and the display of hyperlink tooltips.

I

Id	Describes the position of a point in x, y, z coordinates.
Image	Activates the EXTERNALREFERENCES command.
-Image	Controls the insertion of raster images through the command line.
ImageAdjust	Controls the brightness, contrast, and fading of raster images.
ImageAttach	Attaches raster images to the current drawing.
ImageClip	Places rectangular or irregular clipping boundaries around images.
ImageQuality	Controls the display quality of images.
Import	Imports a variety of file formats into drawings.

Command	Description
Impression	Saves drawings as IRF impression files; available only when the Impression software is installed on your computer.
Imprint	Imprints 2D edges onto the faces of 3D models.
 InitialSetup	Runs the Initial Setup wizard (undocumented).
Insert	Inserts blocks and other drawings into the current drawing.
InsertObj	Inserts objects generated by another Windows application.
Interfere	Determines the interference of two or more 3D solids; displays dialog box with options.
Intersect	Creates a 3D solid or 2D region from the intersection of two or more 3D solids or 2D regions.
Isoplane	F5 and CTRL+E: Switches to the next isoplane.
J	
Join	Unifies individual lines, arcs, splines, and elliptical arcs.
JpgOut	Exports views in JPEG format.
JustifyText	Changes the justification of text.
L	
Layer	Displays the Layer palette; creates and changes layers; toggles the state of layers; assigns linetypes, lineweights, plot styles, colors, and other properties to layers.
LayerClose	Closes the Layer palette.
LayerPalette	Displays the Layer palette.
LAYER	
LayCur	Changes the layer of selected objects to the current layer.
LayDel	Erases all objects from the specified layer; purges layer from drawing.
LayerP	Displays the previous layer state.
LayerPMode	Toggles the availability of the LAYERP command.
LayerState	Displays the Layer State dialog box for creating and editing layer states.
LayFrz	Freezes the layers of the selected objects.
LayIso	Turns off all layers except ones holding selected objects.
LayLck	Locks the layer of the selected object.
LayMch	Changes the layers of selected objects to that of a selected object.
LayMCur	Makes the selected object's layer current, like the AI_MOLC command.
LayMrg	Moves objects to another layer; purges the original layer from the drawing.
LayOff	Turns off the layer of the selected object.
LayOn	Turns on all layers.
LayThw	Thaws all layers.
LayTrans	Translates layer names from one space to another.

Command	Description
LayULk	Unlocks the layer of a selected object.
LayUnIso	Turns on layers that were turned off with the last LAYISO command.
LayVpi	Isolates the selected object's layer in the current viewport.
LayWalk	Displays objects on selected layers.
Layout	Creates a new layout and renames, copies, saves, or deletes existing layouts.
LayoutWizard	Designates page and plot settings for new layouts.
Leader	Draws leader dimensions.
Lengthen	Lengthens or shortens open objects.
Light	Creates, names, places, and deletes “lights” used by the RENDER command.
LightList	Edits light parameters in the Lights In Model palette.
LightListClose	Closes the Lights In Model palette.
Limits	Sets drawing boundaries.
Line	Draws straight line segments.
Linetype	Lists, creates, and modifies linetype definitions; loads them for use in drawings.
List	Displays database information for selected objects.
LiveSection	Activates section planes.
Load	Loads shape files into drawings.
Loft	Creates 3D surfaces and solids by lofting two or more curves.
LogFileOff	Closes the .log keyboard logging file.
LogFileOn	Writes the text of the ‘Command:’ prompt area to .log log file.
LtScale	Specifies the scale factor for linetypes.
LWeight	Sets the current lineweight, lineweight display options, and lineweight units.
M	
Markup	CTRL+7: Reviews and changes the status of marked-up .dwf files.
MarkupClose	CTRL+7: Closes the Markup Set Manager palette.
MassProp	Calculates and displays the mass properties of 3D solids and 2D regions.
MatchCell	Copies the properties from one table cell to other cells.
MatchProp	Copies properties from one object to other objects.
MaterialAttach	Attaches materials to objects by layer name.
MaterialMap	Maps materials to selected objects.
Materials	Manages, applies, and edits materials; opens the Materials palette.
MaterialsClose	Closes the Materials palette.
Measure	Places points or blocks at specified distances along objects.
 MeasureGeom	Measures the distance, radius, angle, area, and volume of selected objects or

Command	Description
	sequence of points.
Menu	Loads .cui and .mnu menu files of AutoCAD commands into the menu area (obsolete).
MenuLoad	Loads .cui and .mnu menu files of AutoCAD commands into the menu area (obsolete).
MenuUnload	Loads .cui and .mnu menu files of AutoCAD commands into the menu area (obsolete).
 Mesh	Creates 3D mesh primitive objects: boxes, cones, cylinders, pyramids, spheres, wedges, and tori.
 MeshCrease	Sharpens the edges of selected mesh subobjects.
 MeshOptions	Displays the Mesh Tessellation Options dialog box, which controls default settings for converting existing objects to mesh objects; previews converted objects.
 MeshPrimitiveOptions	Displays the Mesh Primitive Options dialog box, which sets the tessellation defaults for primitive mesh objects created with the MESH command.
 MeshRefine	Multiplies the number of faces in selected mesh objects or faces.
 MeshSmooth	Converts objects into 3D mesh objects: 3D faces, legacy polygonal meshes, polyface meshes, regions, closed polylines, 3D solids, and 3D surfaces.
 MeshSmoothLess	Decreases the level of smoothness for mesh objects by one level; minimum level is 0.
 MeshSmoothMore	Increases the level of smoothness for mesh objects by one level.
 MeshSplit	Splits a mesh face into two faces.
 MeshUncrease	Removes creases from mesh objects.
MInsert	Inserts arrays of blocks.
Mirror	Creates mirror images of objects.
Mirror3D	Creates mirror images of objects rotated about a plane.
MLeader	Creates multiline leaders.
MLeaderAlign	Aligns two or more mleaders.
MLeaderCollect	Collects two or more mleaders with block annotations into a single mleader with multiple annotations.
MLeaderEdit	Adds and removes leaders from mleaders.
MLeaderStyle	Specifies mleader styles.
MLEdit	Edits multilines.
MLine	Draws multiple parallel lines (up to 16).
MLineStyle	Defines named mline styles, including color, linetype, and endcapping.
Model	Switches from layout tabs to Model tab.
Move	Moves objects.

Command	Description
MRedo	Redoes more than one undo operation.
MSlide	Creates <i>.sld</i> slide files of the current display.
MSpace	Switches to model space.
MtEdit	Edits mtext.
MText	Places formatted paragraph text inside a rectangular boundary.
Multiple	Repeats commands.
MView	Creates and manipulates viewports in paper space.
MvSetup	Sets up new drawings.
N	
NavSMotion	Displays the show motion interface.
NavSMotionClose	Closes the show motion interface.
NavSWheel	SHIFT+W : Displays the navigation steering wheel.
NavVCube	Toggles the display of the ViewCube.
Netload	Loads <i>.net</i> files.
New	CTRL+N : Creates new drawings.
NewSheetSet	Creates a new sheet set.
NewShot	Creates named views with show motion options; displays the Shot Properties tab of the New View / Shot Properties dialog box.
NewView	Shortcut to the VIEW command's New View dialog box.
O	
ObjectScale	Adds and removes scale factors from annotative objects.
Offset	Constructs parallel copies of objects.
OleConvert	Converts OLE (object linking and embedding) objects to other formats (undocumented).
OleLinks	Controls objects linked to drawings.
OleOpen	Opens the source OLE file and application (undocumented).
OleReset	Resets the OLE object to its original form (undocumented).
OleScale	Displays the OLE Properties dialog box.
Oops	Restores objects erased by the previous command.
Open	CTRL+O : Opens existing drawings.
OpenDwfMarkup	Opens marked-up <i>.dwf</i> files.
OpenSheetset	Opens <i>.dst</i> sheet set files.
Options	Customizes AutoCAD's settings.
Ortho	F8 and CTRL+L : Forces lines to be drawn orthogonally or at an angle.
OSnap	F3 and CTRL+F : Locates geometric points of objects.

Command	Description
P	
PageSetup	Specifies the layout page, plotting device, paper size, and settings for new layouts.
Pan	Moves the view within the current viewport.
 Parameters	Defines and edits parameters associated with dimensional constraints through a palette.
 -Parameters	Controls parameters from the command line.
 ParametersClose	Closes the Parameters Manager palette.
PartialLoad	Loads additional geometry into partially-opened drawings.
-PartialOpen	Loads geometry from a selected view or layer into drawings.
PasteAsHyperlink	Pastes objects as hyperlinks.
PasteBlock	CTRL+SHIFT+V: Pastes copied block into drawings.
PasteClip	CTRL+V: Pastes objects from the Clipboard into the upper left corner of drawings.
PasteOrig	Pastes copied objects from the Clipboard into new drawings using the coordinates from the original.
PasteSpec	Controls the format of objects pasted from the Clipboard.
PcInWizard	Imports .pcp and .pc2 configuration files of plot settings.
 PdfAdjust	Adjusts the fade, contrast, and monochrome settings of PDF underlays; can be replaced by the ADJUST super command.
 PdfAttach	Inserts PDF files as an underlay into the current drawing; can be replaced by the ATTACH super command.
 PdfClip	Crops the display of selected PDF underlays to a user-specified boundary; inverts clipping boundaries; can be replaced by the CLIP super command;
 PdfLayers	Toggles the display of layers in PDF underlays; can be replaced by the ULAYERS super command.
PEdit	Edits polylines and polyface objects.
PFace	Constructs polygon meshes defined by the location of vertices.
Plan	Returns to the plan view of the current UCS.
PlaneSurf	Creates flat surface areas.
PLine	Creates connected lines, arcs, and splines of specified width.
Plot	CTRL+P: Plots drawing to printers and plotters.
PlotStamp	Adds information about the drawing to the edge of the plot.
PlotStyle	Sets plot styles for new objects, or assigns plot styles to selected objects.
PlotterManager	Launches the Add-a-Plotter wizard and the Plotter Configuration Editor.
PngOut	Exports views as PNG files (portable network graphics format).
Point	Draws points.
PointLight	Places point lights in drawings.
Polygon	Draws regular polygons with a specified number of sides.

Command	Description
PolySolid	Converts lines, arcs, circles, and polylines into 3D solids with thickness.
PressPull	CTRL+SHIFT+E: Presses and pulls bounded areas.
Preview	Provides a Windows-like plot preview.
Properties	CTRL+1: Displays and changes the properties of existing objects.
PropertiesClose	CTRL+1: Closes the Properties palette.
PSetUpIn	Imports user-defined page setups into new drawing layouts.
PSpace	Switches to paper space (layout mode).
Publish	Plots one or more drawings as a drawing set, or exports them in DWF format.
PublishToWeb	Creates a Web page from one or more drawings in DWF, JPEG, or PNG formats.
Purge	Deletes unused blocks, layers, linetypes, and so on.
Pyramid	Creates 3D solid pyramids and cones.
Q	
QcClose	CTRL+8: Closes the Quick Calc palette.
QDim	Creates continuous dimensions quickly.
QLeader	Creates leaders and leader annotation quickly.
QNew	Starts new drawings based on template files.
QSave	CTRL+S: Saves drawings without requesting a file name.
QSelect	Creates selection sets based on filtering criteria.
QText	Redraws text as rectangles with the next regeneration.
QuickCalc	CTRL+8: Opens the Quick Calc palette.
QuickCui	Opens the Customize User Interface dialog box partially collapsed.
QuickProperties	CTRL+SHIFT+P: Toggles the display of the Quick Properties palette (undocumented).
Quit	ALT+F4 and CTRL+Q: Exits AutoCAD.
QvDrawing	Turns on the thumbnail strip of drawing and layout views.
QvDrawingClose	Closes the thumbnail strip.
QvLayout	Turns on the thumbnail strip of layout views.
QvLayoutClose	Closes the thumbnail strip.
R	
Ray	Draws semi-infinite construction lines.
Recover	Attempts to recover corrupted or damaged files.
RecoverAll	Recovers and updates drawings and all attached reference files.
Rectang	Draws rectangles.
Redefine	Restores AutoCAD's definition of undefined commands.
Redo	CTRL+Y: Restores the operations changed by the previous UNDO command.

Command	Description
Redraw	Cleans up the display of the current viewport.
RedrawAll	Redraws all viewports.
RefClose	Saves or discards changes made during in-place editing of xrefs and blocks.
RefEdit	Selects references for editing.
RefSet	Adds and removes objects from a working set during in-place editing of references.
Regen	Regenerates the drawing in the current viewport.
RegenAll	Regenerates all viewports.
RegenAuto	Controls whether drawings are regenerated automatically.
Region	Creates 2D region objects from existing closed objects.
Reinit	Reinitializes the I/O ports, digitizer, display, plotter, and the <i>acad.pgp</i> file.
Rename	Renames blocks, linetypes, layers, text styles, views, and so on.
Render	Renders 3D objects.
RenderCrop	Renders a rectangular area of the drawing.
RenderEnvironment	Sets the fog options; displays the Render Environment dialog box.
RenderExposure	Changes the brightness and contrast of renderings.
-RenderOutputSize	Presets the size of the next rendering in pixels (undocumented).
RenderPresets	Specifies the parameters for renderings; displays the Render Presets Manager dialog box.
-RenderPresets	Prompts you for rendering settings at the command prompt.
RenderWin	Displays the Render window, which lists and displays the current and previous renderings and parameters.
ResetBlock	Resets dynamic blocks to their original definitions.
Resume	Continues playing a script file that had been interrupted by the ESC key.
RevCloud	Draws revision clouds.
 Reverse	Reverses the order of vertices in lines, polylines, splines, and helixes; the Reverse option is also added to the PEDIT command.
Revolve	Creates 3D solids by revolving 2D closed objects around an axis.
RevSurf	Draws revolved 3D meshes.
Ribbon	Displays the ribbon portion of the user interface.
 +Ribbon	Opens tabs by menugroup name and element id (undocumented).
RibbonClose	Closes the ribbon user interface
Rotate	Rotates objects about specified center points.
Rotate3D	Rotates objects about a 3D axis.
RPref	Sets preferences for renderings; displays the Render Settings palette.
RPrefClose	Closes the Render Settings palette.
RScript	Restarts scripts.

Command	Description
RuleSurf	Draws ruled 3D meshes.
S	
Save / SaveAs	CTRL+SHIFT+S: Saves the current drawing by a specified name.
 +SaveAs	Specifies the format in which to save drawings through the command line: in the 2010 version of DWG, DWT, DWS, or DXF, or in an earlier version as specified by a number (undocumented).
SaveImg	Saves the current rendering in BMP, TGA, or TIF formats.
Scale	Changes the size of objects equally in the x, y, z directions.
ScaleListEdit	Adds and removes scale factors used by PLOT and other commands.
ScaleText	Resizes text.
Script	Runs script files in AutoCAD.
Section	Creates 2D regions from 3D solids by intersecting the solid with a plane.
SectionPlane	Creates section objects independently of the 3D model.
 SectionPlaneJog	Adds a jogged segment to a section object (replaces the JOGSECTION command).
 SectionPlaneSettings	Sets display options for the selected section plane.
 SectionPlaneToBlock	Saves selected section planes as 2D or 3D blocks (replaces the GENERATESECTION command).
SecurityOptions	Sets up passwords and digital signatures for drawings.
 Seek	Opens a Web browser and displays the Autodesk Seek home page for finding blocks and drawings; new name for Content Search and AdCenter's DC Online.
Select	Preselects objects to be edited; CTRL+A selects all objects.
SequencePlay	Plays all show motion views in a view category (undocumented).
SetByLayer	Resets properties to those specified by layers.
SetiDropHandler	Specifies how to treat i-drop objects when dragged into drawings from Web sites.
SetVar	Views and changes AutoCAD's system variables.
-ShadeMode	Shades 3D objects in the current viewport.
Shape	Places shapes from shape files into drawings.
 ShareWithSeek	Uploads blocks and drawings to Autodesk's Seek Web site.
Sheetset	CTRL+4: Opens the Sheet Set Manager palette.
SheetsetHide	CTRL+4: Closes the Sheet Set Manager palette.
Shell	Runs other programs outside of AutoCAD.
ShowMat	Reports the material definition assigned to selected objects.
ShowPalettes	CTRL+SHIFT+H: Shows all palettes that were previously closed with the HIDEPALETTES command.

Command	Description
SigValidate	Displays digital signature information in drawings.
Sketch	Allows freehand sketching.
Slice	Slices 3D solids with a plane.
Snap	F9 and CTRL+B : Toggles snap mode on or off, changes the snap resolution, sets spacing for the X- and Y-axis, rotates the grid, and sets isometric mode.
SolDraw	Creates 2D profiles and sections of 3D solid models in viewports created with the SOLVIEW command.
Solid	Draws filled triangles and rectangles.
SolidEdit	Edits faces and edges of 3D solid objects.
SolProf	Creates profile images of 3D solid models.
SolView	Creates viewports in paper space of orthogonal multi- and sectional view drawings of 3D solid model.
SpaceTrans	Converts length values between model and paper space.
Spell	Checks the spelling of text in the drawing.
Sphere	Draws 3D spheres.
Spline	Draws NURBS (spline) curve.
SplinEdit	Edits splines.
Spotlight	Places spotlights in drawings.
Standards	Compares CAD standards between two drawings.
Status	Displays information about the current drawing.
StlOut	Exports 3D solids to .stl files, in ASCII or binary format, for use with stereolithography.
Stretch	Moves selected objects while keeping connections to other objects unchanged.
Style	Creates and modifies text styles.
StylesManager	Displays the Plot Style Manager.
Subtract	Creates new 3D solids and 2D regions by subtracting one set of objects from a second.
SunProperties	Sets the properties of the Sun light; opens the Sun Properties palette.
SunPropertiesClose	Closes the Sun Properties palette.
Sweep	Creates 3D solids and surfaces by sweeping 2D curves along paths.
SysWindows	CTRL+TAB : Controls the size and position of windows.
T	
Table	Inserts tables in drawings.
TableEdit	Edits the content of table cells.
TableExport	Exports tables as comma-separated text files.
TableStyle	Specifies table styles.

Command	Description
Tablet	F4: Aligns digitizers with existing drawing coordinates; operates only when a digitizer is connected.
TabSurf	Draws tabulated 3D meshes.
TargetPoint	Places target point lights in drawings.
Text	Places text in drawings.
 TextEdit	Edits dimensional constraints, dimensions, and text objects (formerly DDEDIT).
TextScr	F2: Displays the Text window.
TextToFront	Displays text and/or dimensions in front of all other objects.
Thicken	Creates 3D solids by thickening surfaces.
TifOut	Exports views as TIFF files (tagged image file format).
Time	Tracks time.
TInsert	Inserts blocks and drawings in table cells.
Tolerance	Selects tolerance symbols.
Toolbar	Opens the Customize User Interface dialog box (obsolete).
-Toolbar	Controls the display of toolboxes.
ToolPalettes	CTRL+3: Opens the Tool palette.
ToolPalettesClose	CTRL+3: Closes the Tool palette.
Torus	Draws doughnut-shaped 3D solids.
TpNavigate	Sets the default tab for the Tools palette.
Trace	Draws lines with width.
Transparency	Toggles the background of bilevel images between transparent and opaque.
TraySettings	Specifies options for commands operating from the tray (right end of status bar).
TreeStat	Displays information on the spatial index.
Trim	Trims objects by defining other objects as cutting edges.
U	
U	CTRL+Z: Undoes the effect of commands.
UCS	Creates and manipulates user-defined coordinate systems.
UCSicon	Controls the display of the UCS icon.
UcsMan	Manages user-defined coordinate systems.
 ULayers	Controls the display of layers in DWF, DWFx, DGN, and PDF underlays; a super command.
Undefine	Disables commands.
Undo	Undoes several commands in a single operation.
Union	Creates new 3D solids and 2D regions from two solids or regions.
Units	Selects the display format and precision of units and angles.
UpdateField	Updates the contents of field text.

Command	Description
UpdateThumbsNow	Updates the preview images in sheet sets.
V	
VbaIDE	ALT+F8: Launches the Visual Basic Editor.
VbaLoad	Loads VBA projects into AutoCAD.
VbaMan	Loads, unloads, saves, creates, embeds, and extracts VBA projects.
VbaRun	ALT+F11: Runs VBA macros.
VbaStmt	Executes VBA statements at the command prompt.
VbaUnload	Unloads global VBA projects.
View	Saves the display as a view; displays named views.
ViewGo	Displays the named view, along with the associated background setting, layer state, live section, UCS, and visual style. This is a shortcut to the -VIEW command's Restore option.
ViewPlay	Plays the animation associated with a named view.
ViewPlotDetails	Reports on successful and unsuccessful plots.
ViewRes	Controls the fast zoom mode and resolution for circle and arc regenerations.
VisualStyles	Defines and edits visual styles used for renderings; displays the Visual Styles Manager palette.
VisualStylesClose	Closes the Visual Styles Manager palette.
VLisp	Launches the Visual LISP interactive development environment.
VpClip	Clips viewport objects.
VpLayer	Controls the independent visibility of layers in viewports.
VpMax	Maximizes the current viewport (in layout mode).
VpMin	Restores the maximized viewport.
VPoint	Sets the viewpoint from which to view 3D drawings.
VPorts	CTRL+R: Sets the number and configuration of viewports.
VsCurrent	Applies a named visual style to the current viewport.
VSlide	Displays .sld slide files created with the MSLIDE command.
VsSave	Saves a visual style by name in model space only.
VtOptions	Specifies options for visual transitions during the ZOOM command.
W	
WalkFlySettings	Specifies the settings for walk and fly animations; displays the Walk and Fly Settings dialog box.
WBlock	Writes objects to drawing files.
WebLight	Places web lights in drawings; these use .ies files to define light distribution.
Wedge	Creates 3D solid wedges.
WhoHas	Displays ownership information for opened drawing files.

Command	Description
WipeOut	Creates blank areas in drawings.
WmfIn	Imports <i>.wmf</i> files into drawings as blocks.
WmfOpts	Controls how <i>.wmf</i> files are imported.
WmfOut	Exports drawings as <i>.wmf</i> files.
WorkSpace	Creates, modifies, and saves workspaces; sets the current workspace.
WsSave	Saves the current user interface as a workspace.
WsSettings	Specifies options for workspaces through a dialog box.
X	
XAttach	Attaches externally-referenced drawing files to the drawing.
XBind	Binds externally-referenced drawings; converts them to blocks.
XClip	Defines clipping boundaries; sets the front and back clipping planes.
XEdges	Creates wireframe geometry from the edges of 3D solids and surfaces, as well as 2D regions.
XLine	Draws infinite construction lines.
XOpen	Opens externally-referenced drawings in independent windows.
Xplode	Breaks compound objects into component objects, with user control.
-Xref	Places externally-referenced drawings into drawings.
Z	
Zoom	Increases and decreases the viewing size of drawings.
3	
3D	Draws 3D objects of polygon meshes (boxes, cones, dishes, domes, meshes, pyramids, spheres, tori, and wedges).
3dAlign	Functions similar to the ALIGN command, but with different prompts.
3dArray	Creates 3D arrays.
3dClip	Switches to interactive 3D view, and opens the Adjust Clipping Planes palette.
3dConfig	Configures the 3D graphics system in a dialog box.
3dCOrbit	Switches to interactive 3D view, and sets objects into continuous motion.
3dDistance	Switches to interactive 3D view, and makes objects appear closer or farther away.
3dDWF	Exports drawings as 3D DWF files; displays the Export 3D DWF dialog box.
3dDwfPublish	Exports drawings as 3D <i>.dxf</i> files (undocumented).
3dFace	Creates 3D faces.
3dFly	Navigates 3D drawings in fly-through mode; displays the Position Locator palette.
3dFOrbit	Displays an arcball for realtime view changes in 3D.
3dMesh	Draws 3D meshes.
3dMove	Displays the move handle on selected objects, so that they can be moved in 3D

Command	Description
	space.
3dOrbit	Controls the interactive 3D viewing.
3dOrbitCtr	Centers the view.
3dOrbitTransparent	Changes the 3D view in real time (undocumented).
3dPan	Invokes interactive 3D view to drag the view horizontally and vertically.
3dPanTransparent	Pans 3D views in real time (undocumented).
3dPoly	Draws 3D polylines.
 3dPrint	Sends 3D models to 3D printing services in STL format.
3dRotate	Displays the rotate handle on selected objects so that they can be rotated in 3D space.
 3dScale	Scales in 3D, including 3D mesh faces; has a shortcut menu; works with point filters.
3dsIn	Imports 3D Studio geometry and rendering data.
3dSwivel	Switches to interactive 3D view, simulating the effect of turning the camera.
3dSwivelTransparent	Swivels the 3D viewpoint in real time (undocumented).
3dWalk	Enters Walk mode; accesses the Walk Mode right-click menu.
3dZoom	Switches to interactive 3D view to zoom in and out.
3dZoomTransparent	Zooms the 3D view in real time; an undocumented transparent command.

COMMAND ALIASES

Aliases are shortened versions of command names. They are defined in the *acad.ppg* file or “hardwired” by AutoCAD. Those listed below in italics can be used only in the Block Editor. The icon  indicates the 24 aliases that are new to AutoCAD 2010.

Command	Aliases	Command	Aliases
A			
ActRecord	arr	<i>BParameter</i>	<i>param</i>
ActStop	ars	Break	br
-ActStop	-ars	<i>BSave</i>	<i>bs</i>
ActUserInput	aru	<i>BvState</i>	<i>bvs</i>
ActUserMessage	arm	C	
-ActUserMessage	-arm	Camera	cam
AdCenter	adc, dcenter, dc, content		
Align	al	Chamfer	cha
AllPlay	aplay	Change	-ch
AppLoad	ap	CheckStandards	chk
Arc	a	Circle	c
Area	aa	Color	col, colour, ddcolor
Array	ar	CommandLine	cli
-Array	-ar	 ConstraintBar	cbar
AttDef	att, ddattdef	 ConstraintSettings	
-AttDef	-att		csettings
AttEdit	ate, ddatte	Copy	cp, co
-AttEdit	-ate, atte	CTableStyle	ct
AttExt	ddatttext	NavVCube	Cube
AttIpEdit	ati	Cui	toolbar, to, tbconfig
B		CuiLoad	menuload
<i>BAction</i>	<i>ac</i>	CuiUnload	menuunload
<i>BClose</i>	<i>cc</i>	Cylinder	cyl
<i>BParameter</i>	<i>cparam</i>		
BEdit	be	D	
Block	b, bmake, bmod, acadblockdialog	DataExtraction	dx, eatttext
-Block	-b	DataLink	dl
Boundary	bo, bpoly	DataLinkUpdate	dlu
-Boundary	-bo	DbConnect	dbc, ase, aad, aex, asq, ali, aro

Command	Aliases
DdGrips	gr
DdVpoint	vp
 DelConstraint	dcon, delcon
Dist	di
Divide	div
Donut	doughnut, do
DrawingRecovery	drm
Draworder	dr
DSettings	ds, se, ddrmodes, osnap, os, ddosnap
DsViewer	av
DView	dv
DIMENSIONS	
DimAligned	dal, dimali
DimAngular	dan, dimang
DimArc	dar
DimBaseline	dba, dimbase
DimCenter	dce
DimContinue	dco, dimecont
DimDiameter	ddi, dimdia
DimDisassociate	dda
DimEdit	dimed, ded
DimJogged	Jog
DimJogLine	djl
DimLinear	dli, dimlin
DimOrdinate	dor, dimord
DimOverride	dov, dimover
DimRadius	dra, dimrad
DimReassociate	dre
DimStyle	d, dimsty, dst, ddim
DimTEdit	dimted
E	
EditShot	eshot
EditText	ed, ddedit

Command	Aliases
Ellipse	el
Erase	e
Explode	x
Export	exp
 -Export	-qpub
 ExportDwf	qdwf
 ExportDwfx	qdwfx
 ExportPdf	qpdf
-ExportToAutoCAD	aectoacad
Extend	ex
ExternalReferences	er, image, im, xref, xr
Extrude	ext
F	
Fillet	f
Filter	fi
FlatShot	fshot
G	
GeographicLocation	geo, north, northdir
 GeomConstraint	gcon
Gradient	gd
Group	g
-Group	-g
H	
Hatch	bhatch, h, bh
-Hatch	-h
HatchEdit	he
Hide	hi
HidePalettes	poff
I	
-Image	-im
ImageAdjust	iad
ImageAttach	iat
ImageClip	icl

Command	Aliases	Command	Aliases
Import	imp	 MeshSplit	split
Insert	i, inserturl, ddinsert	 MeshUncrease	uncrease
-Insert	-i	Mirror	mi
Insertobj	io	Mirror3D	3dmirror
Interfere	inf	MLeader	mld
Intersect	in	MLeaderAlign	mld
J		MLeaderCollect	mle
Join	j	MLeaderEdit	mle
L		MLeaderStyle	mls
Layer	la, ddlmodes	MLine	ml
-Layer	-la	Move	m
LayerState	las, lman	MSpace	ms
-Layout	lo	MText	t, mt
Leader	lead	-MText	-t
Lengthen	len	MView	mv
Line	l	N	
Linetype	ltype, lt, ddltype	NavSMotion	motion
-Linetype	-ltype, -lt	NavSMotionClose	motioncls
List	ls, li	NavSWheel	wheel
LtScale	lts	NavVCube	cube
Lweight	lineweight, lw	NewShot	nshot
M		NewView	nview
Markup	msm	O	
MatchProp	ma, painter	Offset	o
MaterialMap	setuv	Open	openurl, dxfin
Materials	mat, finish, rmat	Options	op, preferences, ddselect, ddgrips, gr
Measure	me	-OSnap	-os, ddosnap
 MeasureGeom	mea	P	
 MeshCrease	crease	Pan	p
 MeshRefine	refine	-Pan	-p
 MeshSmooth	smooth	-PartialOpen	partialopen
 MeshSmoothLess	less	 Parameters	par
 MeshSmoothMore	more	 -Parameters	-par

Command	Aliases
PasteSpec	pa
PEdit	pe
PLine	pl
Plot	print, dwfout
PlotStamp	ddplotstamp
Point	po
PointLight	freepoint
Polygon	pol
PolySolid	psolid
Preview	pre
Properties	props, pr, mo, ch, ddchprop, ddmodify
PropertiesClose	preclose
PSpace	ps
PublishToWeb	ptw
Purge	pu
-Purge	-pu
Pyramid	pyr
Q	
QLeader	le
QuickCalc	qc
QuickCui	qcui
Quit	exit
QyDrawing	qvd
QyDrawingClose	qvdc
QyLayout	qvl
QyLayoutClose	qvlc
R	
Rectang	rec, rectangle
Redraw	r
RedrawAll	ra
Regen	re
RegenAll	rea
Region	reg

Command	Aliases
Rename	ren
-Rename	-ren
Render	rr
RenderCrop	rc
RenderEnvironment	fog
RenderPresets	rp, rfileopt
RenderWin	rw, rendscr
Revolve	rev
Ribbon	dashboard
RibbonClose	dashboardclose
Rotate	ro
RPref	rpr
S	
Save	saveurl
SaveAs	dxfout
Scale	sc
Script	scr
Section	sec
SectionPlane	splane
 SectionPlaneJog	jogsection
 SectionPlaneToBlock	generatesection
SequencePlay	splay
SetVar	set
ShadeMode	sha
Sheetset	ssm
Shell	sh
ShowPalettes	pon
Slice	sl
Snap	sn
Solid	so
Spell	sp
Spline	spl
SplinEdit	spe
Standards	sta

Command	Aliases	Command	Aliases
Stretch	s	VsCurrent	vs, shademode, sha
Style	st, ddstyle	W	
Subtract	su	Wblock	w, acadwblockdialog
T		-Wblock	-w
Table	tb	Wedge	we
TableStyle	ts	X	
Tablet	ta	XAttach	xa
Text	dt, dtext	XBind	xb
TextEdit	tedit	-XBind	-xb
Thickness	th	XClip	xc
Tilemode	ti, tm	XLine	xl
Tolerance	tol	Xplode	xp
Toolbar	to	-XRef	-xr
ToolPalettes	tp	Z	
Torus	tor	Zoom	z
Tracking	tk, track	3	
Trim	tr	3dAlign	3al
U		3dArray	3a
Ucs	dducs	3dDwf	3dDwfPublish
UcsMan	uc, dducs, dducsp	3dFace	3f
Union	uni	3dMove	3m
Units	un, ddunits	3dOrbit	3do, orbit
-Units	-un	3dPoly	3p
V		 3dPrint	3dp, 3dplot, rapidprototype
View	v, ddview	3dRotate	3r
-View	-v	 3dScale	3s
ViewGo	vgo	3dWalk	3dw, 3dnavigate
ViewPlay	vplay		
VisualStyles	vsm		
-VisualStyles	-vsm		
VLisp	vlide		
VPoint	-vp		
VPorts	viewports		

KEYBOARD SHORTCUTS

Most keyboard shortcuts are customized through the CUI command; some are hardcoded into AutoCAD. The  icon indicates the one shortcut that is revised in AutoCAD 2010.

CONTROL KEYS

<i>Ctrl-key</i>	<i>Meaning</i>
CTRL	Unlocks locked toolbars and windows temporarily; selects a face when picking 3D solid models.
CTRL+0	Toggles clean screen.
CTRL+1	Toggles the Properties palette.
CTRL+2	Toggles the AutoCAD DesignCenter palette.
CTRL+4	Toggles Sheetset Manager palette.
CTRL+6	Launches dbConnect.
CTRL+7	Toggles Markup Set Manager palette.
CTRL+8	Toggles QuickCalc palette.
CTRL+9	Toggles Command Line palette.
CTRL+A	Selects all objects in the current model or layout space.
CTRL+SHIFT+A	Toggles group selection mode.
CTRL+B	Turns snap mode on or off.
CTRL+C	Copies selected objects to the Clipboard.
CTRL+SHIFT+C	Copies selected objects with a base point to the Clipboard.
CTRL+D	Toggles dynamic UCS (formerly toggled coordinate mode on the status bar)
CTRL+E	Switches to the next isoplane.
 CTRL+SHIFT+E	Starts the PRESSPULL command (formerly CTRL+ALT).
CTRL+F	Toggles object snap on and off.
CTRL+G	Turns the grid on and off.
CTRL+H	Toggles pickstyle mode.
CTRL+SHIFT+H	Toggles the visibility of all open palettes (HIDEPALETTES and SHOWPALETTES commands).
CTRL+I	Cycles through coordinate displays (CTRL+D in AutoCAD 2008 and earlier).
CTRL+K	Creates hyperlinks.
CTRL+L	Turns ortho mode on and off.
CTRL+N	Starts new drawings.
CTRL+O	Opens drawings.
CTRL+P	Prints the drawing.
CTRL+SHIFT+P	Toggles Quick Properties palette.
CTRL+Q	Quits AutoCAD.
CTRL+R	Switches to the next viewport.

CTRL+S	Saves the current drawing.
CTRL+SHIFT+S	Displays the Save Drawing As dialog box.
CTRL+T	Toggles tablet mode.
CTRL+V	Pastes from the Clipboard into the drawing or to the command prompt area.
CTRL+SHIFT+V	Pastes with an insertion point.
CTRL+X	Cuts selected objects to the Clipboard.
CTRL+Y	Performs the REDO command.
CTRL+Z	Performs the U command.
CTRL+TAB	Switches to the next drawing.

ALTERNATE KEYS

Alt-key	Meaning
ALT	Accesses keystroke shortcuts on menus.
ALT+TAB	Switches to the next application.

COMMAND LINE KEYSTROKES

These keystrokes are used in the command-prompt area and the Text window.

Keystroke	Meaning
left arrow	Moves the cursor one character to the left.
right arrow	Moves the cursor one character to the right.
HOME	Moves the cursor to the beginning of the line of command text.
END	Moves the cursor to the end of the line.
DEL	Deletes the character to the right of the cursor.
BACKSPACE	Deletes the character to the left of the cursor.
INS	Switches between insert and typeover modes.
up arrow	Displays the previous line in the command history.
down arrow	Displays the next line in the command history.
PGUP	Displays the previous screen of command text.
PGDN	Displays the next screen of command text.
CTRL+V	Pastes text from the Clipboard into the command line.
ESC	Cancels the current command.
TAB	Cycles through command names.

FUNCTION KEYS

<i>Function Key</i>	<i>Meaning</i>
F1	Calls up the help window.
F2	Toggles between the graphics and text windows.
F3	Toggles object snap on and off.
F4	Toggles tablet mode on and off; tablet must first be calibrated.
F5	Switches to the next isometric plane when in iso mode: left, top, and right.
F6	Toggles dynamic UCS on and off.
F7	Toggles grid display on and off.
F8	Toggles ortho mode on and off.
F9	Toggles snap mode on and off; to override temporarily, hold down F9 while selecting objects.
F10	Toggles polar tracking on and off.
F11	Toggles object snap tracking on and off.
F12	Toggles dynamic input.
ALT+F4	Exits AutoCAD.
ALT+F8	Displays the Macros dialog box.
ALT+F11	Starts Visual Basic for Applications editor.
CTRL+F4	Closes the current drawing.

TEMPORARY OVERRIDE (SHIFT) KEYS

Shift-key	Meaning
SHIFT	Toggles orthogonal mode; Selects 3D faces, edges, and vertices; Activates 3DORBIT command; Switches between trim and extend modes during the EXTEND and TRIM commands.
SHIFT+A	Overrides object snap (osnap) mode.
SHIFT+C	Overrides CENter osnap.
SHIFT+D	Disables all snap modes and tracking.
SHIFT+E	Overrides ENDpoint osnap.
SHIFT+J	Overrides osnap tracking mode.
SHIFT+L	Disables all snap modes and tracking.
SHIFT+M	Overrides MIDpoint osnap.
SHIFT+P	Overrides ENDpoint osnap.
SHIFT+Q	Overrides osnap tracking mode.
SHIFT+S	Enables osnap enforcement.
SHIFT+V	Overrides MIDpoint osnap.
SHIFT+W	Toggles the navigation wheel.
SHIFT+X	Overrides polar mode.
SHIFT+'	Overrides osnap mode.
SHIFT+,	Overrides CENter osnap.
SHIFT+.	Overrides polar mode.
SHIFT+;	Enables osnap enforcement.

MOUSE AND DIGITIZER BUTTONS

Button #	Mouse Button	Meaning
...	Wheel	Zooms or pans.
1	Left	Selects objects.
3	Center, wheel	Displays object snap menu.
2	Right	Displays shortcut menus.
4		Cancels command.
5		Toggles snap mode.
6		Toggles orthographic mode.
7		Toggles grid display
8		Toggles coordinate display.
9		Switches to isometric plane.
10		Toggles tablet mode.
SHIFT+1	SHIFT+Left	Toggles cycle mode.
SHIFT+2	SHIFT+Left	Displays object snap shortcut menu.
CTRL+2	CTRL+Right	Displays object snap shortcut men

MTEXT EDITOR SHORTCUT KEYS

Shortcut	Meaning
TAB	Moves cursor to the next tab stop, or creates the next indent level.
SHIFT+TAB	Outdents.
CTRL+A	Selects all text.
CTRL+B	Boldfaces.
CTRL+I	<i>Italicizes.</i>
CTRL+U	<u>Underlines.</u>
CTRL+C	Copies selected text to the Clipboard.
CTRL+X	Cuts text from the editor and sends it to the Clipboard.
CTRL+V	Pastes text from the Clipboard.
CTRL+Y	Redoes last undo.
CTRL+Z	Undoes last action.
CTRL+F	Inserts field text; displays Field dialog box.
CTRL+R	Finds and replaces text.
CTRL+SPACE	Removes formatting from selected text.
CTRL+SHIFT+SPACE	Inserts non-breaking space.
CTRL+SHIFT+U	Converts selected text to all UPPERCASE.
CTRL+SHIFT+L	Converts selected text to all lowercase.

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AutoCAD System Variables

AutoCAD stores information about its current state, the drawing and the operating system in over 400 *system variables*. Those variables help programmers — who often work with menu macros and AutoLISP — to determine the state of the AutoCAD system.

CONVENTIONS

The following pages list all documented system variables, plus several more not documented by Autodesk. The listing uses the following conventions:

Bold	System variable is documented by Autodesk in AutoCAD 2010.
<i>Bold Italic</i>	System variable is listed neither by the SETVAR command nor by Autodesk's documentation.
	System variable must be accessed through the SETVAR command; otherwise, the equivalent command is executed.
Default	System variable's default value, as set in the <i>acad.dwg</i> prototype drawing; other template files may have different default values.
(R/O)	System variable is read-only (cannot be changed by the user).
Toggle	System variable has one of two values: 0 (off, closed, disabled) or 1 (on, open, enabled).
ACI color	System variable takes on one of AutoCAD's 255 colors; ACI is short for "AutoCAD color index."
	System variable is new to AutoCAD 2010.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
 _AuthorPalettePath	<i>varies</i>	<i>Path to the customized Tool palette folder.</i>
_PkSer (R/O)	<i>varies</i>	<i>Software serial number, such as "117-69999999".</i>
_Server (R/O)	0	Network authorization code.
 _ToolPalettePath	<i>varies</i>	<i>Path to the Tool palette folder.</i>
_VerNum (R/O)	<i>varies</i>	<i>Internal program build number, such as "Z.54.01".</i>

A

AcadLspAsDoc	0	Controls whether <i>acad.lsp</i> is loaded into: 0 The first drawing only. 1 Every drawing.
AcadPrefix (R/O)	<i>varies</i>	Specifies paths used by AutoCAD search in Options Files dialog box.
AcadVer (R/O)	"18.0"	Specifies the AutoCAD version number.
AcisOutVer	70	Controls the ACIS version number; values are 15, 16, 17, 18, 20, 21, 30, 40, or 70.
AcGiDumpMode	0	<i>Value of 0 or 1.</i>
ActPath	""	Specifies paths to action macro files.
ActRecorderState	0	Reports the status of the Action Recorder: 0 Inactive. 1 Actively recording a macro. 2 Actively playing back a macro.
ActRecPath	"C:\users ..."	Specifies current path to recorded action macros.
ActUi	6	Controls state of Active Macro panel on the ribbon: 0 No change. 1 Expands during playback. 2 Expands during recording. 4 Prompts for name and description when recording is complete.
AdcState (R/O)	0	Toggle: reports if DesignCenter is active.
AFlags	0	Controls the default attribute display mode: 0 No mode specified. 1 Invisible. 2 Constant. 4 Verify. 8 Preset. 16 Lock position in block. 32 Multiple-line attributes.
AngBase	0	Controls the direction of zero degrees relative to the UCS.
AngDir	0	Controls the rotation of positive angles: 0 Clockwise. 1 Counterclockwise.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
AnnoAllVisible	1	Toggles display of annotative objects at the current scale: 0 Displays only objects matching VP scale. 1 Displays all annotative objects (default).
AnnoAutoScale	4	Updates annotative objects when the annotation scale is changed: 1 Adds the new annotation scale to annotative objects (except on layers that are off, frozen, locked, or have viewports set to freeze). 2 As above, but includes objects on locked layers (excludes objects on layers that are off, frozen, or have viewports set to freeze). 3 The opposite of 2: applies to annotative objects on all layers, except objects on locked layers. 4 Applies to all objects regardless of layer status. -1 Same as 1, but turned off. -2 Same as 2, but turned off. -3 Same as 3, but turned off. -4 Same as 4, but turned off.
AnnotativeDwg	0	Toggles whether the drawing acts like an annotative block when inserted into other drawings: 0 Nonannotative block. 1 Annotative block. (This sysvar is read-only when drawing contains annotative text.)
ApBox	0	Toggles display of the AutoSnap aperture box cursor.
 Aperture	10	Controls the object snap aperture in pixels: 1 Minimum size. 50 Maximum size.
AppFrameResources  (R/O)		<i>Specifies resource files used by applications.</i> <i>"pack:/ / application:,,, / AcWindows; etc..."</i>
ApState	0	Toggle: reports the state of the ApBox variable.
 Area (R/O)	0.0	Reports the area measured by the last Area command.
AttDia	0	Controls the user interface for entering attributes: 0 Command-line prompts. 1 Dialog box.
AttIpe	0	Toggles display of in-place attribute text editor's toolbar: 0 Reduced-function toolbar. 1 Full-function toolbar.
AttMode	1	Controls the display of attributes: 0 Off. 1 Normal. 2 On; displays invisible attributes.
AttMulti	1	Toggles creation of multi-line attributes: 0 Attributes can be single-line only. 1 Attributes can be multi-line.
AttReq	1	Toggles attribute values during insertion: 0 Uses default values. 1 Prompts user for values.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
AuditCtl	0	Toggles creation of <i>.adt</i> audit log files: 0 File not created. 1 File created.
AUnits	0	Controls the type of angular units: 0 Decimal degrees. 1 Degrees-minutes-seconds. 2 Grads. 3 Radians. 4 Surveyor's units.
AUPrec	0	Specifies number of decimal places displayed by angles; range is 0 - 8.
AutoDwfPublish	1	Toggles automatic publishing of DWF files when drawings are saved or closed (to be removed in a future release): 0 DWF are not published. 1 DWF are published.
AutomaticPub 	0	Toggles automatic exporting of drawing as DWF and PDF files when drawings are saved or closed: 0 DWF and PDF files are not published. 1 DWF and PDF are published.
AutoSnap	63	Controls the AutoSnap display (sum): 0 Turns off all AutoSnap features. 1 Turns on marker. 2 Turns on SnapTip. 4 Turns on magnetic cursor. 8 Turns on polar tracking. 16 Turns on object snap tracking. 32 Turns on tooltips for polar tracking and object snap tracking.
B		
BackgroundPlot	2	Controls background plotting and publishing (ignored during scripts): 0 Plot foreground; publish foreground. 1 Plot background; publish foreground. 2 Plot foreground; publish background. 3 Plot background; publish background.
BackZ	0.0	Controls the location of the back clipping plane offset from the target plane.
BActionBarMode 	1	Toggles between action objects and action bars in the Block Editor: 0 Displays legacy action objects (compatible with AutoCAD 2009 and earlier). 1 Displays action bars.
BActionColor	"7"	Specifies ACI text color for actions in Block Editor.
BConStatusMode 	0	Toggles the display of the constraint status.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
BDependencyHighlight	1	Toggles highlighting of dependent objects when parameters, actions, or grips selected in Block Editor: 0 Not highlighted. 1 Highlighted.
BGripObjColor	"141"	Specifies ACI color of grips in Block Editor.
BGripObjSize	8	Controls the size of grips in Block Editor; range is 1 to 255.
BgrdPlotTimeout	20	<i>Controls the timeout for failed background plots; ranges from 0 to 300 seconds.</i>
BindType	0	Controls how xref names are converted when being bound or edited: 0 From <i>xref\name</i> to <i>xref\$0\$name</i> . 1 From <i>xref\name</i> to <i>name</i> .
 BlipMode	0	Toggles the display of blip marks.
BlockEditLock	0	Toggles the locking of dynamic blocks being edited: 0 Unlocked. 1 Locked.
BlockEditor (R/O)	0	Reports whether Block Editor is open.
BlockTestWindow  (R/O)	0	Reports whether the Block Test window is open.
BParameterColor	"7"	Specifies ACI color of parameters in the Block Editor.
BParameterFont	"Simplex.shx"	Controls the font used for parameter and action text in the Block Editor.
BParameterSize	12	Controls the size of parameter text and features in Block Editor: 1 Minimum. 255 Maximum.
BPTextHorizontal 	1	Toggles the alignment of text with dimension lines in Block Editor: 0 Text is aligned with dimension lines. 1 Text is horizontal.
BTMarkDisplay	1	Controls the display of value set markers: 0 Unlocked. 1 Locked.
BVMode	0	Controls the display of invisible objects in the Block Editor: 0 Invisible. 1 Visible and dimmed.
C		
CalcInput	1	Controls how formulas and global constants are evaluated in dialog boxes: 0 Not evaluated. 1 Evaluated after pressing Alt+Enter.
CameraDisplay	0	Toggles the display of camera glyphs.
CameraHeight	0	Specifies the default height of cameras.
CAnnoScale	"1:1"	Names the current annotative scale for the current viewport.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
CAnnoScaleValue (R/O)	1.0	Reports the current annotative scale.
CaptureThumbnails	1	Determines how thumbnails are generated by NavSWheel's Restore option: 0 No preview thumbnails generated . 1 Generated when bracket is positioned over empty frames. 2 Generated automatically after every view change.
CBarTransparency 	50	<i>Controls the transparency of constraint bars; range is 10 to 90.</i>
CConstraintForm 	0	Determines the type of new constraint applied to objects by the DimConstraint command: 0 Applies dynamic constraints. 1 Applies annotative constraints.
CDate (R/O)	<i>varies</i>	Specifies the current date and time in the format YyyyMmDd.HhMmSsDd, such as 20080503.18082328
CDynDisplayMode	0	<i>Toggles display of dynamic display.</i>
CeColor	"BYLAYER"	Controls the current color.
CeLtScale	1.0	Controls the current linetype scaling factor.
CeLType	"BYLAYER"	Controls the current linetype.
CeLWeight	-1	Controls the current lineweight in millimeters; valid values are 0, 5, 9, 13, 15, 18, 20, 25, 30, 35, 40, 50, 53, 60, 70, 80, 90, 100, 106, 120, 140, 158, 200, and 211, plus the following: -1 BYLAYER. -2 BYBLOCK. -3 Default, as defined by LwDdefault.
CenterMT	0	Controls how corner grips stretch uncentered multiline text: 0 Center grip moves in same direction; opposite grip stays in place. 1 Center grip stays in place; side grips move in direction of stretch.
ChamferA	0.0	Specifies the current value of the first chamfer distance.
ChamferB	0.0	Specifies the current value of the second chamfer distance.
ChamferC	0.0	Specifies the current value of the chamfer length.
ChamferD	0	Specifies the current value of the chamfer angle.
ChamMode	0	Toggles the chamfer input mode: 0 Chamfer by two lengths. 1 Chamfer by length and angle.
CipMode	0	<i>Toggles Customer Involvement Program.</i>
CircleRad	0.0	Specifies the most-recent circle radius.
CLayer	"0"	Specifies name of current layer.
CleanScreenState (R/O)	0	Toggle: reports whether cleanscreen mode is active.
CliState (R/O)	1	Reports the command line palette.
CMaterial	"ByLayer"	Sets the name of the current material.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
CmdActive (R/O)	1	Reports the type of command currently active (used by programs): 1 Regular command. 2 Transparent command. 4 Script file. 8 Dialog box. 16 Dynamic data exchange. 32 AutoLISP command. 64 ARX command.
CmdDia	1	Toggles QLeader's inplace text editor.
CmdEcho	1	Toggles AutoLISP command display: 0 No command echoing. 1 Command echoing.
CmdInputHistoryMax	20	Controls the maximum command input items stored; works with InputHistoryMode.
CmdNames (R/O)	<i>varies</i>	Reports name of the command currently active, such as "SETVAR".
CMLleaderStyle	"Standard"	Reports name of current mleader style.
CMLJust	0	Controls the multiline justification mode: 0 Top. 1 Middle. 2 Bottom.
CMLScale	1.0	Controls the scale of overall multiline width: -n Flips offsets of multiline. 0 Collapses to single line. n Scales by a factor of <i>n</i>.
CMLStyle	"STANDARD"	Specifies the current multiline style name.
Compass	0	Toggles the display of the 3D compass.
ConstraintBarDisplay 	1	Toggles the display of constraint bars.
ConstraintBarMode 	4031	Determines the geometric constraints displayed by constraint bars: 1 Horizontal 2 Vertical 4 Perpendicular 8 Parallel 16 Tangent 32 Smooth 64 Coincident 128 Concentric 256 Colinear 512 Symmetric 1024 Equal 2048 Fix 4031 All displayed.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
ConstraintNameFormat 	2	Determines how constraint text is displayed: 0 By variable names. 1 By values. 2 By expressions (formulas).
ConstraintRelax 	0	Toggles constraints between enforced and relaxed during editing.
ConstraintSolveMode 	3	Determines how constraints behave when applied or edited: 0 Does not infer additional constraints. 1 Infers temporary constraints, when applied. 2 Infers temporary constraints, when edited by grips.
Coords	1	Controls the coordinate display style: 0 Updated by screen picks. 1 Continuous display. 2 Polar display upon request.
CopyMode	0	Toggles whether the Copy command repeats itself: 0 Command repeats copying. 1 Command makes one copy, then exits.
CPlotStyle	"ByColor"	Specifies the current plot style; options for named plot styles are: ByLayer, ByBlock, Normal, and User Defined.
CProfile (R/O)	"<<Unnamed Profile>>"	Specifies the name of the current profile.
CrossingAreaColor	100	ACI color of crossing rectangle.
CShadow	0	Specifies how shadows are cast by 3D objects (if graphics board is capable; see 3dConfig command): 0 Casts and receives shadows. 1 Casts shadows. 2 Receives shadows. 3 Ignores shadows.
CTab	"Model"	Specifies the name of the current tab.
CTableStyle	"Standard"	Specifies the name of the current table style name.
CursorSize	5	Controls the cursor size as a percent of the viewport size. 1 Minimum size. 100 Full viewport.
CVPport	2	Specifies the current viewport number.
D		
DataLinkNotify	2	Controls reporting on data links: 0 All notifications disabled. 1 Displays data link icon in tray. 2 Displays the icon and a warning balloon in the tray.
Date (R/O)	<i>varies</i>	Reports the current date in Julian format, such as 2448860.54043252
DbcState (R/O)	0	Toggle: specifies whether dbConnect Manager is active.
DblClkEdit	1	Toggles editing by double-clicking objects.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
DBMod (R/O)	0	Reports how the drawing has been modified: 0 No modification since last save. 1 Object database modified. 2 Symbol table modified. 4 Database variable modified. 8 Window modified. 16 View modified. 32 Field modified.
DctCust	"sample.cus"	Specifies the name of custom spelling dictionary.
DctMain	"enu"	Controls the code for spelling dictionary: enu American English eng British English (ise) enc Canadian English cat Catalan csy Czech dan Danish nld Dutch (primary) fin Finnish fra French (accented capitals) frc French (unaccented capitals) deu German (post-reform) deo German (pre-reform) ita Italian nor Norwegian (Bokmal) ptb Portuguese (Brazilian) ptg Portuguese (Iberian) rus Russian esp Spanish sve Swedish
DefaultGizmo 	0	Determines which gizmo is the default when subobjects are selected: 3D Move, 3D Rotate, 3D Scale, or none.
DefaultIndex 	0	Range is 0 to 255.
DefaultLighting	1	Toggles distant lighting.
DefaultLightingType	0	Toggles between new (1) and old (0) type of lights.
DefaultViewCategory	""	Specifies the default name for View Category in the View command's New View dialog box.
DefLPIStyle	"ByColor"	Reports the default plot style for layer 0.
DefPISyle	"ByColor"	Reports the default plot style for new objects.
DelObj	1	Toggles the deletion of source objects: -2 Users are prompted whether to erase all defining objects. -1 Users are prompted whether to erase profiles and cross sections. 0 Objects retained. 1 Profiles and cross sections erased. 2 All defining objects erased.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
DemandLoad	3	Controls application loading when drawing contains proxy objects: 0 Apps not demand-loaded. 1 Apps loaded when drawing opened. 2 Apps loaded at first command. 3 Apps loaded when drawing is opened or at first command.
DgnFrame	0	Controls display of DGN underlay frame: 0 Does not display frame. 1 Displays and plots frame. 2 Displays but does not plot frame.
DgnImportMax	1000000	Limits maximum MicroStation elements to import into AutoCAD; 0 = no limit.
DgnMappingPath (R/O)	"C:\users\..."	Path to folder holding the dgnsetups.ini file.
DgnOsnap	1	Toggles object snapping to DGN elements: 0 Does not osnap. 1 Osnaps.
DiaStat (R/O)	0	Reports whether user exited dialog box by clicking: 0 Cancel button. 1 OK button.

Dimension Variables

DimADec	0	Controls angular dimension precision: -1 Use DimDec setting (default). 0 Zero decimal places (minimum). 8 Eight decimal places (maximum).
DimAlt	Off	Toggles alternate units: On Enabled. Off Disabled.
DimAltD	2	Controls alternate unit decimal places.
DimAltF	25.4	Controls alternate unit scale factor.
DimAltRnd	0.0	Controls rounding factor of alternate units.
DimAltTD	2	Controls decimal places of tolerance alternate units; range is 0 to 8.
DimAltTZ	0	Controls display of zeros in alternate tolerance units: 0 Zeros not suppressed. 1 All zeros suppressed. 2 Include 0 feet, but suppress 0 inches. 3 Includes 0 inches, but suppress 0 feet. 4 Suppresses leading zeros. 8 Suppresses trailing zeros.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
DimAltU	2	Controls display of alternate units: 1 Scientific. 2 Decimal. 3 Engineering. 4 Architectural; stacked. 5 Fractional; stacked. 6 Architectural. 7 Fractional. 8 Windows desktop units setting.
DimAltZ	0	Controls the display of zeros in alternate units: 0 Suppresses 0 ft and 0 in. 1 Includes 0 ft and 0 in. 2 Includes 0 ft; suppress 0 in. 3 Suppresses 0 ft; include 0 in. 4 Suppresses leading 0 in dec dims. 8 Suppresses trailing 0 in dec dims. 12 Suppresses leading and trailing zeroes.
DimAnno (R/O)	0	Reports whether current dimstyle is: 0 Not annotative. 1 Annotative.
DimAPost	""	Specifies the prefix and suffix for alternate text.
DimArcSym	0	Specifies the location of the arc symbol: 0 Before dimension text. 1 Above the dimension text. 2 Not displayed.
DimAso	On	<i>Toggles associative dimensions:</i> On Dimensions are created associative. Off Dimensions are not associative.
DimAssoc	2	Controls how dimensions are created: 0 Dimension elements are exploded. 1 Single dimension object, attached to defpoints. 2 Single dimension object, attached to geometric objects.
DimASz	0.18	Controls the default arrowhead length.
DimAtFit	3	Controls how text and arrows are fitted when there is insufficient space between extension lines (leader is added when DimTMove = 1): 0 Text and arrows outside extension lines. 1 Arrows first outside, then text. 2 Text first outside, then arrows. 3 Either text or arrows, whichever fits better.

Variable Name	Default Value	Meaning
DimAUnit	0	Controls the format of angular dimensions: 0 Decimal degrees. 1 Degrees.Minutes.Seconds. 2 Grads. 3 Radians. 4 Surveyor units.
DimAZin	0	Controls the display of zeros in angular dimensions: 0 Displays all leading and trailing zeros. 1 Suppresses 0 in front of decimal. 2 Suppresses trailing zeros behind decimal. 3 Suppresses zeros in front and behind the decimal.
DimBlk	""	Specifies the name of the arrowhead block: Architectural tick: Box filled: Box: Closed blank: Closed filled: Closed: Datum triangle filled: Datum triangle: Dot blanked: Dot small: Dot: Integral: None: Oblique: Open 30: Open: Origin indication: Right-angle: "Archtick" "Boxfilled" "Boxblank" "Closedblank" "" (default) "Closed" "Datumfilled" "Datumblank" "Dotblank" "Dotsmall" "Dot" "Integral" "None" "Oblique" "Open30" "Open" "Origin" "Open90"
DimBlk1	""	Specifies the name of first arrowhead's block; uses same list of names as under DimBlk. . No arrowhead.
DimBlk2	""	Specifies name of second arrowhead block.
DimCen	0.09	Controls how center marks are drawn: -n Draws center lines. 0 No center mark or lines drawn. +n Draws center marks of length n.
DimClrD	0	Specifies ACI color of dimension lines: 0 BYBLOCK (default). 256 BYLAYER.
DimClrE	0	Specifies ACI color of extension lines and leaders.
DimClrT	0	Specifies ACI color of dimension text.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
DimConstraintIcon 	2	Toggles the display of the lock icon next to dimensional constraints: 0 Does not display the lock icon. 1 Displays icon next to dynamic constraints. 2 Displays the icon next to annotational constraints. 3 Displays the icon next to dynamic and annotational constraints.
DimDec	4	Controls the number of decimal places displayed; range is 0 to 8.
DimDLE	0.0	Controls the length of the dimension line extension.
DimDLI	0.38	Controls the increment of the continued dimension lines.
DimDSEP	","	Specifies the decimal separator (must be a single character).
DimExe	0.18	Controls the extension above the dimension line.
DimExO	0.0625	Specifies the extension line origin offset.
DimFrac	0	Controls the fraction format when DimLUnit set to 4 or 5: 0 Horizontal. 1 Diagonal. 2 Not stacked.
DimFXL	1	Specifies default length of fixed extension lines.
DimFxlOn	0	Toggles fixed extension lines.
DimGap	0.09	Controls the gap between text and the dimension line.
DimJogAngle	45	Specifies default angle for jogged dimension lines.
DimJust	0	Controls the positioning of horizontal text: 0 Center justify. 1 Next to first extension line. 2 Next to second extension line. 3 Above first extension line. 4 Above second extension line.
DimLdrBlk	""	Specifies the name of the block used for leader arrowheads; same as DimBlock. . Suppresses display of arrowhead.
DimLFac	1.0	Controls the linear unit scale factor.
DimLim	Off	Toggles the display of dimension limits.
DimLtEx1	""	Specifies linetype for the first extension line.
DimLtEx2	""	Specifies linetype for second extension line.
DimLtype	""	Specifies linetype name for dimension line.
DimLUnit	2	Controls dimension units (except angular); replaces DimUnit: 1 Scientific. 2 Decimal. 3 Engineering. 4 Architectural. 5 Fractional. 6 Windows desktop.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
DimLwD	-2	Controls the dimension line lineweight; valid values are BYLAYER, BYBLOCK, or integer multiples of 0.01mm.
DimLwE	-2	Controls the extension lineweight.
DimPost	""	Specifies the default prefix or suffix for dimension text (maximum 13 characters): "" No suffix.
DimRnd	0.0	Controls the rounding value for dimension distances.
DimSAh	Off	Toggles separate arrowhead blocks: Off Use arrowheads defined by DimBlk. On Use arrowheads defined by DimBlk1 and DimBlk2.
DimScale	1.0	Controls the overall dimension scale factor: 0 Value is computed from the scale between current model space viewport and paper space. >0 Scales text and arrowheads.
DimSD1	Off	Toggles display of the first dimension line: On First dimension line is suppressed. Off Not suppressed.
DimSD2	Off	Toggles display of the second dimension line: On Second dimension line is suppressed. Off Not suppressed.
DimSE1	Off	Toggles display of the first extension line: On First extension line is suppressed. Off Not suppressed.
DimSE2	Off	Toggles display of the second extension line: On Second extension line is suppressed. Off Not suppressed.
DimSho	On	Toggles dimension updates while dragging: On Dimensions are updated during drag. Off Dimensions are updated after drag.
DimSOXD	Off	Toggles display of dimension lines outside of extension lines: On Dimension lines not drawn outside extension lines. Off Dimension lines are drawn outside extension lines.
 DimStyle (R/O)	"STANDARD"	Reports the current dimension style.
DimTAD	0	Controls the vertical position of text: 0 Centers text between extension lines. 1 Places text above dimension line, except when dimension line not horizontal and DimTIH = 1. 2 Places text on side of dimension line farthest from the defining points. 3 Conforms to JIS (Japanese Industry Standard).
DimTDec	4	Specifies number of decimal places for primary tolerances; range 0 - 8.
DimTFac	1.0	Controls the scale factor for tolerance text height.

Variable Name	Default Value	Meaning
DimTFill	0	Toggles background fill color for dimension text.
DimTFillClr	0	Specifies background color for dimension text.
DimTIH	On	Toggles alignment of text placed inside extension lines: Off Text aligned with dimension line. On Text is horizontal.
DimTIX	Off	Toggles placement of text inside extension lines: Off Text placed inside extension lines, if room. On Text forced between the extension lines.
DimTM	0.0	Controls the value of the minus tolerance.
DimTMove	0	Controls how dimension text is moved: 0 Moves dimension line with text. 1 Moves text, and adds a leader to the text. 2 Move text anywhere without leader.
DimTOFL	Off	Toggles placement of dimension lines: Off Dimension lines not drawn when arrowheads are outside. On Dimension lines drawn, even when arrowheads are outside.
DimTOH	On	Toggles text alignment when outside of extension lines: Off Text aligned with dimension line. On Text is horizontal.
DimTol	Off	Toggles generation of dimension tolerances: Off Tolerances not drawn. On Tolerances are drawn.
DimTolJ	1	Controls vertical justification of tolerances: 0 Bottom. 1 Middle. 2 Top.
DimTP	0.0	Specifies the value of the plus tolerance.
DimTSz	0.0	Controls the size of oblique tick strokes: 0 Arrowheads. >0 Oblique strokes.
DimTVP	0.0	Controls the vertical position of text when DimTAD = 0: 1 Turns on DimTAD (=1). >-0.7 or <0.7 Splits dimension line for text.
DimTxSty	"STANDARD"	Specifies the dimension text style.
DimTxt	0.18	Controls the text height.
DimTxtDirection 	Off	Toggles the reading direction of dimension text: 0 Left to right. 1 Right to left.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
DimTZin	0	Controls the display of zeros in tolerances: 0 Suppresses 0 ft and 0 in. 1 Includes 0 ft and 0 in. 2 Includes 0 ft; suppress 0 in. 3 Suppresses 0 ft; include 0 in. 4 Suppresses leading 0 in decimal dim. 8 Suppresses trailing 0 in decimal dim. 12 Suppresses leading and trailing zeroes.
DimUPT	Off	Controls user-positioned text: Off Cursor positions dimension line. On Cursor also positions text.
DimZIN	0	Controls the display of zero in feet-inches units: 0 Suppresses 0 ft and 0 in. 1 Includes 0 ft and 0 in. 2 Includes 0 ft; suppress 0 in. 3 Suppresses 0 ft; include 0 in. 4 Suppresses leading 0 in decimal dim. 8 Suppresses trailing 0 in decimal dim. 12 Suppresses leading and trailing zeroes.
DispSilh	0	Toggles the silhouette display of 3D solids.
Distance (R/O)	0.0	Reports the distance last measured by the Dist command.
DivMeshBoxHeight 	3	Specifies the default number of subdivisions along the height of boxes; range is 1 to 256.
DivMeshBoxLength 	3	Specifies the default number of subdivisions along the length of boxes; range is 1 to 256.
DivMeshBoxWidth 	3	Specifies the default number of subdivisions along the width of boxes; range is 1 to 256.
DivMeshConeAxis 	8	Specifies the default number of subdivisions around the base of cones; range is 1 to 256.
DivMeshConeBase 	3	Specifies the default number of subdivisions on the base of cones; range is 1 to 256.
DivMeshConeHeight 	3	Specifies the default number of subdivisions between the base and the tip of cones; range is 1 to 256.
DivMeshCylAxis 	8	Specifies the default number of subdivisions around the trunk of cylinders; range is 1 to 256.
DivMeshCylBase 	3	Specifies the default number of subdivisions on the endcaps of cylinders; range is 1 to 256.
DivMeshCylHeight 	3	Specifies the default number of subdivisions along the trunk of cylinders.
DivMeshPyrBase 	3	Specifies the default number of subdivisions on the base of pyramids; range is 1 to 256.
DivMeshPyrHeight 	3	Specifies the default number of subdivisions between the base and the tip of pyramids; range is 1 to 256.

Variable Name	Default Value	Meaning
DivMeshPyrLength 	3	Specifies the default number of subdivisions along the base of pyramids; range is 1 to 256.
DivMeshSphereAxis 	12	Specifies the default number of subdivisions between the two polar ends of spheres; range is 1 to 256.
DivMeshSphereHeight 	6	Specifies the default number of subdivisions around spheres; range is 1 to 256.
DivMeshTorusPath 	8	Specifies the default number of subdivisions along paths swept by tori profiles; range is 1 to 256.
DivMeshTorusSection 	8	Specifies the default number of subdivisions around paths swept by tori profiles; range is 1 to 256.
DivMeshWedgeBase 	3	Specifies the default number of subdivisions on the base of wedges; range is 1 to 256.
DivMeshWedgeHeight 	3	Specifies the default number of subdivisions along the height of wedges; range is 1 to 256.
DivMeshWedgeLength 	4	Specifies the default number of subdivisions along the length of wedges; range is 1 to 256.
DivMeshWedgeSlope 	3	Specifies the default number of subdivisions along the slope of wedges; range is 1 to 256.
DivMeshWedgeWidth 	3	Specifies the default number of subdivisions along the width of wedges; range is 1 to 256.
DonutId	0.5	Controls the inside diameter of donuts.
DonutOd	1.0	Controls the outside diameter of donuts.
 DragMode	2	Controls the drag mode: 0 No drag. 1 On if requested. 2 Automatic.
DragP1	10	Controls the regen drag display; range is 0 to 32767.
DragP2	25	Controls the fast drag display; range is 0 to 32767.
DragVs	""	Specifies the default visual style when 3D solids are created by dragging the cursor; disabled when visual style is 2D wireframe.
DrawOrderCtrl	3	Controls the behavior of draw order: 0 Draw order not restored until next regen or drawing reopened. 1 Normal draw order behavior. 2 Draw order inheritance. 3 Combines options 1 and 2.
DrState	0	Toggles the Drawing Recovery palette.
DTextEd	2	Controls the user interface of DText/Text command: 0 Uses in-place text editor. 1 Displays Edit Text dialog box. 2 Click elsewhere in drawing to start new text string.
DwfFrame	2	Determines display of the frame around DWF overlays: 0 Frame is turned off. 1 Frame is displayed and plotted. 2 Frame is displayed but not plotted.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
DwfOsnap	1	Toggles osnapping of the DWF frame.
DwgCheck	0	Toggles checking of whether drawing was edited by software other than AutoCAD: 0 Suppresses dialog box. 1 Displays warning dialog box. 2 Prints warning in command bar.
DwgCodePage (R/O)	<i>varies</i>	Same values as SysCodePage.
DwgName (R/O)	<i>varies</i>	Reports the current drawing file name, such as "drawing1.dwg".
DwgPrefix (R/O)	<i>varies</i>	Reports the drawing's drive and folder, such as "d:\acad 2010\".
DwgTitled (R/O)	0	Reports whether the drawing file name is: 0 "drawing1.dwg". 1 User-assigned name.
DxEval	12	Controls which commands cause AutoCAD to check for changed external data files (sum): 0 None. 1 Open. 2 Save. 4 Plot. 8 Publish. 12 Plot and Publish. 16 eTransmit and Archive. 32 Save with automatic update. 64 Plot with automatic update. 128 Publish with automatic update. 256 eTransmit/Archive with automatic update.
DynConstraintDisplay 	1	Toggles the display of dynamic constraints.
DynConstraintMode 	1	Toggles the display of hidden constraints: 0 Constraints remain hidden when objects are selected. 1 Constraints are displayed.
DynDiGrip	31	Controls the dynamic dimensions displayed during grip stretch editing (DynDiVis =2): 0 Displays none. 1 Resulting dimension. 2 Length change dimension. 4 Absolute angle dimension. 8 Angle change dimension. 16 Arc radius dimension. 31 Displays all.
DynDiVis	1	Controls dynamic dimensions displayed during grip stretch editing: 0 First (in the cycle order). 1 First two (in the cycle order). 2 All (as specified by DynDiGrip).

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
DynMode	3	Controls dynamic input features. (Click DYN on status bar to turn on hidden modes.) -3 Both on hidden. -2 Dimensional input on hidden. -1 Pointer input on hidden. 0 Off. 1 Pointer input on. 2 Dimensional input on. 3 Both on.
DynPiCoords	0	Toggles pointer input coordinates: 0 Relative. 1 Absolute.
DynPiFormat	0	Toggles pointer input coordinates: 0 Polar. 1 Cartesian.
DynPiVis	1	Controls when pointer input is displayed: 0 When user types at prompts for points. 1 When prompted for points. 2 Always.
DynPrompt	1	Toggles display of prompts in Dynamic Input tooltips.
DynToolTips	1	Toggles which tooltips are affected by tooltip appearance settings: 0 Only Dynamic Input value fields. 1 All drafting tooltips.
E		
EdgeMode	0	Toggles edge mode for the Trim and Extend commands: 0 Does not extend. 1 Extends cutting edge.
Elevation	0.0	Specifies the current elevation, relative to current UCS.
EnterpriseMenu (R/O)	""	Reports the path and .cui file name.
EntExts	1	Controls how drawing extents are calculated: 0 Extents calculated every time; slows down AutoCAD, but uses less memory. 1 Extents of every object are cached as a two- byte value (default). 2 Extents of every object are cached as a four- byte value (fastest but uses more memory).
ErrNo (R/O)	0	Reports error numbers from AutoLISP, ADS, and Arx.
ErState (R/O)	0	Reports on the display of the External References palette.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
Expert	0	Controls the display of prompts: 0 Normal prompts. 1 ‘About to regen, proceed?’ and ‘Really want to turn the current layer off?’ 2 ‘Block already defined. Redefine it?’ and ‘A drawing with this name already exists. Overwrite it?’ 3 Linetype command messages. 4 UCS Save and VPorts Save. 5 DimStyle Save and DimOverride.
ExplMode	1	Toggles whether the Explode and Xplode commands explode non-uniformly scaled blocks: 0 Does not explode. 1 Explodes.
ExportEPlotFormat 	2	Specifies the format in which to export drawings: 0 PDF. 1 DWF. 2 DWFx.
ExportModelSpace 	0	Specifies which part of model space to export in DWF/x or PDF format: 0 Drawing as currently displayed. 1 Extents of the drawing. 2 Windowed area.
ExportPageSetup 	0	Specifies which page setup to use in exporting drawings in DWF/x or PDF format: 0 Default setup. 1 Overridden setup.
ExportPaperSpace 	0	Specifies which part of paper space export in DWF/x or PDF format: 0 Current layout only. 1 All layouts.
ExtMax (R/O)	-1.0E+20, -1.0E+20, -1.0E+20	Upper-right coordinate of drawing extents.
ExtMin (R/O)	1.0E+20, 1.0E+20, 1.0E+20	Lower-left coordinate of drawing extents.
ExtNames	1	Controls the format of named objects: 0 Names are limited to 31 characters, and can include A - Z, 0 - 9, dollar (\$), underscore (_), and hyphen (-). 1 Names are limited to 255 characters, and can include A - Z, 0 - 9, spaces, and any characters not used by Windows or AutoCAD for special purposes.
F		
FaceterDevNormal 	40	Specifies the maximum angle between adjacent mesh surfaces and surface normals; range is any positive angle.
FaceterDevSurface 	0.0010	Specifies the accuracy of meshes converted from surfaces and solids; range is any positive number: 0 Turns off this option.

Variable Name	Default Value	Meaning
FaceterGridRatio 	0.0000	Specifies the maximum aspect ratio for mesh subdivisions; range is 0 to 100. Values <1 are inverted: $0.5 = 2$.
FaceterMaxEdgeLength 	0.0000	Specifies the maximum edge length for meshes converted from surfaces and solids; range is any positive number.
FaceterMaxGrid 	4096	Specifies the maximum grid lines in the U and V directions for meshes converted from surfaces and solids; range is 0 to 4096.
FaceTerMeshType 	0	Specifies the type of mesh created from solids and surfaces: 0 Optimized mesh type. 1 Quadrilateral type. 2 Triangular type.
FaceterMinUGrid 	0	Specifies the minimum number of grid lines in the U direction for meshes converted from solids and surfaces; range is 0 to 1023: 0 Turns off this option.
FaceTerMinVGrid 	0	Specifies the minimum number of grid lines in the V direction for meshes converted from solids and surfaces; range is 0 to 1023: 0 Turns off this option.
FaceterPrimitiveMode 	1	Toggles smoothness settings for meshes converted from solids and surfaces: 0 Applies settings from the Mesh Tessellation Options dialog box. 1 Applies settings from the Mesh Primitive Options dialog box.
FaceterSmoothLev 	1	Specifies the level of smoothness for meshes converted from solids and surfaces: <0 Does not smooth objects. 0 Does not smooth objects. 1 Applies smoothness level 1. 2 Applies smoothness level 2. 3 Applies smoothness level 3.
FacetRatio	0	Controls the aspect ratio of facets on rounded 3D bodies: 0 Creates an n by 1 mesh. 1 Creates an n by m mesh.
FaceTRres	0.5000	Controls the smoothness of shaded and hidden-line objects; range is 0.01 to 10.
FieldDisplay	1	Toggles background to field text: 0 No background. 1 Gray background.
FieldEval	31	Controls how fields are updated: 0 Not updated. 1 Updated with Open. 2 Updated with Save. 4 Updated with Plot. 8 Updated with eTransmit. 16 Updated with regeneration.
FileDia	1	Toggles user interface for file-access commands, like Open and Save: 0 Displays command-line prompts. 1 Displays file dialog boxes.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
FilletRad	0.0	Specifies the current fillet radius.
FillMode	1	Toggles the fill of solid objects, wide polylines, fills, and hatches.
FontAlt	"simplex.shx"	Specifies the font used for missing fonts.
FontMap	"acad.fmp"	Specifies the name of the font mapping file.
Force_Paging	0	<i>Ranges from 0 to 4.29497E9.</i>
Frame 	3	Toggles the visibility of frames on all xrefs, images, and DWF, DWFX, PDF, and DGN underlays; overrides the XClipFrame, ImageFrame, DwfFrame, PdfFrame, DgnFrame commands: 0 Not visible; not plotted 1 Visible; plotted. 2 Visible; not plotted. 3 Uses the settings of the XClipFrame, ImageFrame, DwfFrame, PdfFrame, DgnFrame commands.
FrontZ (R/O)	0.0	Reports the front clipping plane offset.
FullOpen (R/O)	1	Reports whether the drawing is: 0 Partially loaded. 1 Fully open.
FullPlotPath	1	Specifies the format of file name sent to plot spooler: 0 Drawing file name only. 1 Full path and drawing file.
G		
GeoLatLongFormat	0	Specifies format of latitude and longitude: 0 Decimal degrees. 1 Degrees, minutes, and seconds.
GeoMarkerVisibility	1	Toggles visibility of geographic markers.
GfAng	0	<i>Controls the angle of gradient fill; 0 to 360 degrees.</i>
GfClr1	"RGB 000,000,255"	<i>Specifies the first gradient color in RGB format.</i>
GfClr2	"RGB 255,255,153"	<i>Specifies the second gradient color in RGB format.</i>
GfClrLum	1.0	<i>Controls the level of gray in one-color gradients:</i> 0 Black. 1 White.
GfClrState	1	<i>Specifies the type of gradient fill:</i> 0 Two-color. 1 One-color.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
GfName	1	Specifies the style of gradient fill: 1 Linear. 2 Cylindrical. 3 Inverted cylindrical. 4 Spherical. 5 Inverted spherical. 6 Hemispherical. 7 Inverted hemispherical. 8 Curved. 9 Inverted curved.
GfAShift	0	Controls the origin of the gradient fill: 0 Centered. 1 Shifted up and left.
GlobCheck	0	Controls reporting on dialog boxes: -1 Turns off local language. 0 Turns off. 1 Warns if larger than 640x400. 2 Reports size in pixels. 3 Displays additional information.
GridDisplay	2	Determines grid display (sum of bitcodes): 0 Grid restricted to area specified by the Limits command. 1 Grid is infinite. 2 Adaptive grid, with fewer grid lines when zoomed out. 4 Displays more grid lines when zoomed in. 8 Grid follows the xy plane of the dynamic UCS.
GridMajor	5	Specifies number of minor grid lines per major line.
GridMode	0	Toggles the display of the grid.
GridUnit	0.5,0.5	Controls the x, y spacing of the grid.
GripBlock	0	Toggles the display of grips in blocks: 0 At block insertion point. 1 Of all objects within block.
GripColor	150	Specifies ACI color of unselected grips.
GripDynColor	140	Specifies ACI color of grips in dynamic blocks.
GripHot	12	Specifies ACI color of selected grips.
GripHover	11	Specifies ACI grip color when cursor hovers.
GripObjLimit	100	Controls the maximum number of grips displayed; range is 1 to 32767; 0 = grips never suppressed.
Grips	1	Toggles the display of grips.
GripSize	5	Specifies the size of grip; range 1 - 255 pixels.
GripSizeMesh 	3	Specifies the size of grips used for mesh objects; range 1 - 255 pixels.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
GripSubObjMode 	1	Toggles the use of Shift to select subobjects.
GripTips	1	Toggles display of grip tips when cursor hovers over custom objects.
GtAuto	1	Toggles display of grip tools.
GtDefault	0	Toggles which commands are the default commands in 3D views: 0 Scale , Move, and Rotate. 1 3dScale , 3dMove, and 3dRotate.
GtLocation	0	Controls location of grip tools: 0 Aligns grip tool with UCS icon. 1 Aligns with the last selected object.
H		
HaloGap	0	Controls the distance by which haloed lines are shortened in 2D wireframe visual style; a percentage of 1 unit.
Handles (R/O)	On	Reports whether object handles (not grips) can be accessed by applications; obsolete.
HidePrecision	0	Controls the precision of hide calculations in 2D wireframe visual style: 0 Single precision, less accurate, faster. 1 Double precision, more accurate, but slower (recommended).
HideText	On	Controls the display of text during Hide: On Text is neither hidden nor hides other objects, unless text object has thickness. Off Text is hidden and hides other objects.
HideXrefScales	1	<i>Toggles inclusion of xrefs in the list of scale factors.</i>
Highlight	1	Toggles object selection highlighting.
HPAng	0.0	Specifies current hatch pattern angle.
HpAssoc	1	Toggles associativity of hatches: 0 Not associative. 1 Associative.
HpBound	1	Controls the object created by the Hatch and Boundary commands: 0 Region. 1 Polyline.
HpDouble	0	Toggles double hatching.
HpDrawOrder	3	Controls draw order of hatches and fills: 0 None. 1 Behind all other objects. 2 In front of all other objects. 3 Behind the hatch boundary. 4 In front of the hatch boundary.
HpGapTol	0	Controls largest gap allowed in hatch boundaries; ranges from 0 to 5000 units.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
HpInherit	0	Toggles how MatchProp copies the hatch origin from source object to destination objects: 0 As specified by HpOrigin. 1 As specified by the source hatch object.
HpMaxlines	1000000	Controls the maximum number of hatch lines; range is 100 - 10,000,000.
HpName	"ANSI31"	Specifies default hatch name.
HpObjWarning	10000	Specifies the maximum number of hatch boundaries that can be selected before AutoCAD flashes warning message; range is 1 to 1073741823.
HpOrigin	0,0	Specifies the default origin for hatch objects.
HpOriginMode	0	Controls the default hatch origin point: 0 Specified by HpOrigin. 1 Bottom-left corner of hatch's rectangular extents. 2 Bottom-right corner of rectangular extents. 3 Top-right corner of rectangular extents. 4 Top-left corner of rectangular extents. 5 Center of hatch's rectangular extents.
HpScale	1.0	Specifies the current hatch scale factor; cannot be zero.
HpSeparate	0	Controls number of hatch objects made from multiple boundaries: 0 Single hatch object created. 1 Separate hatch object created.
HpSpace	1.0	Controls the default spacing of user-defined hatches; cannot be zero.
HyperlinkBase	""	Specifies the path for relative hyperlinks.
I		
ImageFrame	1	Toggles the visibility of frames on images: 0 Not visible; not plotted 1 Visible; plotted. 2 Visible; not plotted.
ImageHlt	0	Toggles image frame highlighting when raster images are selected.
ImpliedFace	1	Toggles detection of implied faces.
IndexCtl	0	Controls creation of layer and spatial indices: 0 No indices created. 1 Layer index created. 2 Spatial index created. 3 Both indices created.
InetLocation	Specifies the default URL for Browser. "\"http://www.autodesk.com\""	

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
InputHistoryMode	15	Controls the content and location of user input history (bitcode sum of): 0 No history displayed. 1 Displayed at the command line, and in dynamic prompt tooltips accessed with Up and Down arrow keys. 2 Current command displayed in the shortcut menu. 4 All commands displayed in the shortcut menu. 8 Blipmark for recent input displayed in the drawing. 15 All on.
InsBase	0.0,0.0,0.0	Controls the default insertion base point relative to the current UCS for Insert and XRef commands.
InsName	""	Specifies the default block name: . Set to no default.
InsUnits	1	Specifies drawing units of blocks dragged into drawings: 0 Unitless. 1 Inches. 2 Feet. 3 Miles. 4 Millimeters. 5 Centimeters. 6 Meters. 7 Kilometers. 8 Microinches. 9 Mils. 10 Yards. 11 Angstroms. 12 Nanometers. 13 Microns. 14 Decimeters. 15 Decameters. 16 Hectometers. 17 Gigameters. 18 Astronomical Units. 19 Light Years. 20 Parsecs.
InsUnitsDefSource	1	Controls source drawing units value; ranges from 0 to 20; see above.
InsUnitsDefTarget	1	Controls target drawing units; see list above.
IntelligentUpdate	20	Controls graphics refresh rate in frames per second; range is 0 (off) to 100 fps.
InterfereColor	"1"	Specifies color of interference objects.
InterfereObjVs	"Realistic"	Specifies visual style of interference objects.
InterfereVpVs	"3d wireframe"	Specifies visual style during interference checking.

Variable Name	Default Value	Meaning
IntersectionColor	257	Specifies ACI color of intersection polylines in 2D wireframe visual style (257=ByEntity).
IntersectionDisplay	Off	Toggles display of 3D surface intersections during Hide command in 2D wireframe: Off Does not draw intersections. On Draws polylines at intersections.
ISaveBak	1	Toggles creation of <i>.bak</i> backup files.
ISavePercent	50	Controls the percentage of waste in saved <i>.dwg</i> file before cleanup occurs: 0 Slower full saves. >0 Faster partial saves. 100 Maximum.
IsoLines	4	Controls the number of contour lines on 3D solids; range is 0 - 2047.
L		
LargeObjectSupport 	0	Toggles support for large-object support in DWG files: 0 Create objects compatible with AutoCAD 2009 and earlier. 1 Use large objects in drawings.
LastAngle (R/O)	0	Reports the end angle of last-drawn arc.
LastPoint	0,0,0	Reports the x, y, z coordinates of the last-entered point.
LastPrompt (R/O)	"varies"	Reports the last string on the command line.
Latitude	"37.7950"	Specifies last-used angle of latitude.
LayerDlgMode	1	Toggles the command mapped to the Layer command: 0 ClassicLayer command. 1 LayerPalette command.
LayerEval	1	Controls when newly-added (unreconciled) layers are detected: 0 Never. 1 When xref layers are added to the drawing 2 When new layers are added to drawings and xrefs.
LayerEvalCtl	1	Controls unreconciled layer filters: 0 Disables notification of new layers. 1 Enables notification, as controlled by the LayerEval sysvar.
LayerFilterAlert	2	Controls the deletion of layer filters in excess of 99 filters <i>and</i> the number of layers: 0 Off. 1 Deletes all filters without warning, when layer dialog box opened. 2 Recommends deleting all filters when layer dialog box opened. 3 Displays dialog box for selecting filters to erase, upon opening the drawing.
LayerManagerState (R/O)	0	Reports on the status of the Layer Properties Manager palette.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
LayerNotify	15	Controls when AutoCAD displays alerts about unreconciled layers (bitcode): 0 Off. 1 Plotting. 2 Opening drawings. 4 Loading, reloading, and attaching xrefs. 8 Restoring layer states. 16 Saving drawings. 32 Inserting blocks, etc.
LayLockFadeCtl	90	Controls the fading of locked layers during the LayIso command: 0 Not faded. >0 Faded up to 90 percent. <0 Not faded, but the value is saved.
LayoutRegenCtl	2	Controls display list for layouts: 0 Display-list regen'ed with each tab change. 1 Display-list is saved for model tab and last layout tab. 2 Display list is saved for all tabs.
LegacyCtrlPick	0	Toggles function of Ctrl+pick: 0 Selects faces, edges, and vertices of 3D solids. 1 Cycles through overlapping objects.
LensLength (R/O)	50.0	Reports perspective view lens length, in mm.
LightGlyphDisplay	1	Toggles display of light glyph.
LightingUnits	2	Controls the type of lighting used: 0 Generic lighting. 1 International units of photometric lighting. 2 American units of photometric lighting.
LightListState	0	Toggles display of Light List palette.
LightsInBlocks	1	Toggles use of lights in blocks: 0 Off. 1 On.
LimCheck	0	Toggles drawing limits checking.
LimMax	12.0,9.0	Controls the upper right drawing limits.
LimMin	0.0,0.0	Controls the lower left drawing limits.
LinearBrightness	0	Controls the overall brightness of generic lighting in renderings; range is -10 to 10.
LinearContrast	0	Controls the overall contrast of generic lighting in renderings; range is -10 to 10.
LispInit	1	Toggles AutoLISP functions and variables: 0 Preserved from drawing to drawing. 1 Valid in current drawing only.
Locale (R/O)	"enc"	Reports ISO language code; see DctMain.
LocalRootPrefix (R/O)	"d:\docume..."	Reports the path to folder holding local customizable files.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
LockUi	0	Controls the position and size of toolbars and palettes; hold down Ctrl key to unlock temporarily (bitcode sum): 0 Toolbars and palettes unlocked. 1 Docked toolbars locked. 2 Docked palettes locked. 4 Floating toolbars locked. 8 Floating palettes locked.
LoftAng1	90	Specifies angle of loft to first cross-section; range is 0 to 359.9 degrees.
LoftAng2	90	Specifies angle of loft to second cross-section.
LoftMag1	0.0	Specifies magnitude of loft at first cross-section; range is 1 to 10.
LoftMag2	0.0	Specifies magnitude of loft at last cross-section.
LoftNormals	1	Specifies location of loft normals: 0 Ruled. 1 Smooth. 2 First normal. 3 Last normal. 4 Ends normal. 5 All normal. 6 Use draft angle and magnitude.
LoftParam	7	Specifies the loft shape: 1 Minimizes twists between cross-sections. 2 Aligns start-to-end direction of each cross-section. 4 Generates simple solids and surfaces, instead of spline solids and surfaces. 8 Closes the surface or solid between the first and last cross-sections.
LogExpBrightness	65.0	Controls the overall brightness of photometric lighting in renderings; range is 0 to 200.
LogExpContrast	50.0	Controls the overall contrast of photometric lighting in renderings; range is 0 to 100.
LogExpDaylight	2	Controls how daylight is displayed photometric renderings: 0 Off. 1 On. 2 Same as sun status.
LogExpMidtones	1.00	Controls the overall midtones of photometric lighting in renderings; range is 0.01 to 20.
LogExpPhysicalScale	1500.000	Scales photometric lights physically.
LogFileMode	0	Toggles writing command prompts to <i>.log</i> file.
LogFileName (R/O)	"...\Drawing1.log"	Reports file name and path for <i>.log</i> file.
LogFilePath	"d:\acad 2010\"	Specifies path to the <i>.log</i> file.
LogInName (R/O)	"username"	Reports user's login name; truncated after 30 characters.
Longitude	-122.3940	Specifies current angle of longitude (default = San Francisco).
 LTScale	1.0	Controls linetype scale factor; cannot be 0.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
LUnits	2	Controls linear units display: 1 Scientific. 2 Decimal. 3 Engineering. 4 Architectural. 5 Fractional.
LUPrec	4	Controls decimal places (or inverse of smallest fraction) of linear units; range is 0 to 8.
LwDefault	25	Controls the default lineweight, in millimeters; must be one of the following values: 0, 5, 9, 13, 15, 18, 20, 25, 30, 35, 40, 50, 53, 60, 70, 80, 90, 100, 106, 120, 140, 158, 200, or 211.
LwDisplay	0	Toggles whether lineweights are displayed; setting saved separately for Model space and each layout tab.
LwUnits	1	Toggles units used for lineweights: 0 Inches. 1 Millimeters.
M		
MacroTrace	0	<i>Toggles diesel debug mode.</i>
MatState	0	Toggles display of Materials palette.
MaxActVP	64	Controls the maximum number of viewports to display; range is 2 to 64.
MaxObjMem	0	<i>Controls the maximum number of objects in memory; object pager is turned off when value = 0, <0, or 2,147,483,647.</i>
MaxSort	1000	Controls the maximum names sorted alphabetically; range 0 - 32767.
MButtonPan	1	Toggles the behavior of the wheel mouse: 0 As defined by AutoCAD .cui file. 1 Pans when dragging with wheel.
MeasureInit	0	Toggles drawing units for default drawings: 0 English. 1 Metric.
Measurement	0	Toggles current drawing units: 0 English. 1 Metric.
MenuBar	0	Toggles the display of the menu bar.
MenuCtl	1	Toggles the display of submenus in side menu: 0 Only with menu picks. 1 Also with keyboard entry.
MenuEcho	0	Controls menu and prompt echoing (sum): 0 Displays all prompts. 1 Suppresses menu echoing. 2 Suppresses system prompts. 4 Disables ^P toggle. 8 Displays all input-output strings.

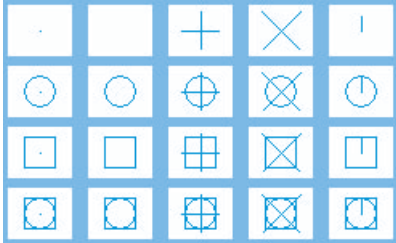
<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
MenuName (R/O)	"acad"	Reports path and file name of .cui file.
MeshType 	1	Toggles the type of mesh used by the RevSurf, TabSurf, RuleSurf, EdgeSurf, and other commands: 0 Polyfaces (compatible with AutocAD 2009 and earlier). 1 3D mesh objects (AutoCAD 2010 and later).
MirrText	0	Toggles text handling by Mirror command: 0 Retains text orientation. 1 Mirrors text.
MLeaderScale	1	Scales mleaders based on: 0.0 Makes scale the ratio of model space viewport to paper space. >0 Specifies scale factor.
ModeMacro	""	Invokes Diesel macros.
MsLtScale	1	Controls how linetypes are scaled: 0 Not scaled by the annotation scale. 1 Scaled by the annotation scale.
MsmState (R/O)	0	Specifies if Markup Set Manager is active: 0 No. 1 Yes.
MsOleScale	1.0	Controls the size of text-containing OLE objects when pasted in model space: -1 Scales by value of PlotScale. 0 Scales by value of DimScale. >0 Scale factor.
MTextEd	"Internal"	Controls the name of the MText editor: . Uses the default editor. 0 Cancels the editing operation. -1 Uses the secondary editor. "blank" Uses MText internal editor. "Internal" Uses MText internal editor. "oldeditor" Uses the previous internal editor. "Notepad" Uses Windows Notepad editor. ":lispd" Uses Built-in AutoLISP function. <i>string</i> Uses another editor with a name fewer than 256 characters long in the form of this syntax: :<i>AutoLISP</i>textEditorFunction#<i>TextEditor</i>.
MTextFixed	2	Controls the mtext editor appearance: 0 Mtext editor is used. 1 Mtext editor remembers its location. 2 Difficult-to-read text is displayed horizontally at a larger size.
MTJigString	"abc"	Specifies the sample text displayed by mtext editor; maximum 10 letters; enter . for no text.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
MTextToolbar	1	Toggles toolbar display during MText: 0 Does not display toolbar. 1 Displays Text Formatting toolbar from AutoCAD 2008 and earlier. 2 Displays ribbon's Text Formatting panel.
MyDocumentsPrefix (R/O)		Reports path to \my documents folder of the logged-in user. "C:\Documents and Settings\username\My Documents"
N		
NavSWheelMode	2	Specifies style of steering wheel: 0 Big wheel for viewing objects. 1 Big wheel for touring buildings. 2 Big wheel for full navigation. 3 Big wheel for 2D navigation. 4 Mini wheel for viewing objects. 5 Mini wheel for touring buildings. 6 Mini wheel for full navigation.
NavSWheelOpacityBig	50	Changes translucency of big steering wheel. Range is 25% to 90% (almost opaque).
NavSWheelOpacityMini	150	Changes translucency of mini steering wheel. Range is 25% to 90% (almost opaque).
NavSWheelSizeBig	1	Specifies size of the big steering wheel: 0 Small size. 1 Normal size. 2 Large size.
NavSWheelSizeMini	1	Specifies size of the mini steering wheel: 0 Small size. 1 Normal size. 2 Large size. 3 Extra large.
<i>NavSWheelWalkSpeed</i>	<i>1.0</i>	<i>Sets speed of Walk mode; range is 0.1 to 10.</i>
NavVCubeDisplay	0	Toggles display of the viewcube.
NavVCubeLocation	0	Locates viewcube in current viewport: 0 Upper-right corner. 1 Upper-left corner. 2 Lower-left corner. 3 Lower-right corner.
NavVCubeOpacity	50	Sets translucency of viewcube; 0 = displayed only when cursor passes over it.
NavVCubeOrient	1	Specifies viewcube's orientation: 0 Relative to WCS. 1 Relative to current UCS.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
NavVCubeSize	1	Sizes the viewcube: 0 Small size. 1 Normal size. 2 Large size.
NodeName (R/O)	"AC\$"	<i>Reports the name of the network node; range is one to three characters.</i>
NoMutt	0	Toggles display of messages (aka muttering) during scripts, LISP, macros: 0 Displays prompt, as normal. 1 Suppresses muttering.
NorthDirection	0	Specifies angle of sun relative to positive y axis, and N direction of viewcube.
NwfState	1	<i>Display New Features Workshop at AutoCAD start.</i>
O		
ObscuredColor	257	Specifies ACI color of objects obscured by Hide in 2D wireframe visual style.
ObscuredLtype	0	Specifies linetype of objects obscured by Hide in 2D wireframe visual mode: 0 Invisible. 1 Solid. 2 Dashed. 3 Dotted. 4 Short dash. 5 Medium dash. 6 Long dash. 7 Double short dash. 8 Double medium dash. 9 Double long dash. 10 Medium long dash. 11 Sparse dot.
OffsetDist	-1.0	Controls current offset distance: <0 Offsets through a specified point. ≥0 Default offset distance.
OffsetGapType	0	Controls how polylines reconnect when segments are offset: 0 Extends segments to fill gap. 1 Fills gap with fillet (arc segment). 2 Fills gap with chamfer (line segment).
OleFrame	2	Controls visibility of OLE frames: 0 Frame is not displayed and not plotted. 1 Frame is displayed and is plotted. 2 Frame is displayed but is not plotted.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
OleHide	0	Controls display and plotting of OLE objects: 0 All OLE objects visible. 1 Visible in paper space only. 2 Visible in model space only. 3 Not visible.
OleQuality	3	Specifies quality of display and plotting of embedded OLE objects: 0 Monochrome. 1 Low quality graphics. 2 High quality graphics. 3 Automatically selected mode.
OleStartup	0	Toggles loading of OLE source applications to improve plot quality: 0 Does not load OLE source application. 1 Loads OLE source app when plotting.
OpenPartial	1	Determines whether drawings can be edited when not fully loaded: 0 Drawings must be fully open. 1 Visible portions of drawings can be edited before file is fully open.
OpmState	0	Toggles whether Properties palette is active.
OrthoMode	0	Toggles orthographic mode.
OsMode	4133	Controls current object snap mode (sum): 0 NONE. 1 ENDpoint. 2 MIDpoint. 4 CENter. 8 NODe. 16 QUAdrant. 32 INTersection. 64 INSertion. 128 PERpendicular. 256 TANgent. 512 NEARest. 1024 QUIck. 2048 APPint. 4096 EXTension. 8192 PARallel. 16383 All modes on. 16384 Object snap turned off via OSNAP on the status bar.
OSnapCoord	2	Controls keyboard overrides object snap: 0 Object snap overrides keyboard. 1 Keyboard overrides object snap. 2 Keyboard overrides object snap, except in scripts.

Variable Name	Default Value	Meaning
OSnapHatch	0	Toggles whether hatches are snapped: 0 Osnaps ignore hatches. 1 Hatches are snapped.
OSnapNodeLegacy	1	Toggles whether osnap snaps to mtext insertion points.
OsnapZ	0	Toggles osnap behavior in z direction: 0 Uses the z-coordinate. 1 Uses the current elevation setting.
OsOptions	3	Determines when objects with negative z values are osnapped: 0 Uses the actual z coordinate. 1 Substitutes z coordinate with the elevation of the current UCS.
P		
PaletteOpaque	0	Controls transparency of palettes: 0 Turned off by user. 1 Turned on by user. 2 Unavailable, but turned on by user. 3 Unavailable, and turned off by user.
PaperUpdate	0	Toggles how AutoCAD plots layouts with paper size different from plotter's default: 0 Displays a warning dialog box. 1 Changes paper size to that of the plotter configuration file.
ParameterCopyMode 	1	Determines how constraints and referenced variables are copied: 0 Does not copy constraints. 1 Copies constraints and variables; expressions are constants. 2 Copies constraints, variables, and expressions; references variables; replaces data with constants. 3 Copies constraints, variables, and expressions; references variables; adds variables, if necessary. 4 Copies all; adds variables, if necessary.
ParametersStatus  (R/O)	0	Reports whether the Parameters Manager palette is open: 0 Closed. 1 Open.
PdfFrame 	1	Toggles the display of the PDF frame for editing and plotting: 0 Not displayed; not plotted. 1 Displayed; plotted. 2 Displayed; not plotted.
PdfOsnap 	1	Toggles osnapping between frame and objects in attached PDF files: 0 Geometry not osnapped. 1 Geometry osnapped.

Variable Name	Default Value	Meaning
PDMode	0	Controls point display style (sum):  0 Dot. 1 No display. 2 +-symbol. 3 x-symbol. 4 Short line. 32 Circle. 64 Square.
PDSize	0.0	Controls point display size: >0 Absolute size, in pixels. 0 5% of drawing area height. <0 Percentage of viewport size.
PEditAccept	0	Toggles display of the PEdit command's 'Object selected is not a polyline. Do you want to turn it into one? <Y>:' prompt.
PEllipse	0	Toggles object used to create ellipses: 0 True ellipses. 1 Polyline arcs.
Perimeter (R/O)	0.0	Reports perimeter calculated by Area, DbList, and List commands.
Perspective	0	Toggles perspective mode; not available in 2D wireframe visual mode.
PerspectiveClip	5.0	Controls position of eyepoint clipping, as a percentage; range is 0.01% - 10.0%.
PFaceVMax (R/O)	4	Reports the maximum vertices per 3D face.
PickAdd	1	Toggles meaning of Shift key on selection sets: 0 Adds to selection set. 1 Removes from selection set.
PickAuto	1	Toggles selection set mode: 0 Single pick mode. 1 Automatic windowing and crossing.
PickBox	3	Controls selection pickbox size; range is 0 to 50 pixels.
PickDrag	0	Toggles selection window mode: 0 Pick two corners. 1 Pick a corner; drag to second corner.
PickFirst	1	Toggles command-selection mode: 0 Enter command first. 1 Select objects first.
PickStyle	1	Controls how groups and associative hatches are selected: 0 Includes neither. 1 Includes groups. 2 Includes associative hatches. 3 Includes both.
Platform (R/O)	"varies"	Reports the name of the operating system.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
PLineConvertMode 	0	Determines how polylines are converted to splines: 0 Linear segments. 1 Arc segments.
PLineGen	0	Toggles polyline linetype generation: 0 From vertex to vertex. 1 From end to end.
PLineType	2	Controls automatic conversion and creation of 2D polylines by PLine: 0 Not converted; creates old-format polylines. 1 Not converted; creates optimized lwpolylines. 2 Polylines in older drawings are converted on open; PLine creates optimized lwpolyline objects.
PLineWid	0.0	Controls current polyline width.
PlotOffset	0	Toggles the plot offset measurement: 0 Relative to edge of margins. 1 Relative to edge of paper.
PlotRotMode	2	Controls the orientation of plots: 0 Lower left = 0,0. 1 Lower left plotter area = lower left of media. 2 X, y-origin offsets calculated relative to the rotated origin position.
PlQuiet	0	Toggles display during batch plotting and scripts (replaces CmdDia): 0 Displays plot dialog boxes and nonfatal errors. 1 Logs nonfatal errors; does not display plot dialog boxes.
PolarAddAng	""	Holds a list of up to 10 user-defined polar angles; each angle can be up to 25 characters long, each separated with a semicolon (;). For example: 0;15;22.5;45.
PolarAng	90	Controls the increment of polar angle; contrary to Autodesk documentation, you can specify any angle.
PolarDist	0.0	Controls the polar snap increment when SnapStyl is set to 1 (isometric).
PolarMode	0	Controls polar and object snap tracking: 0 Measures polar angles based on current UCS (absolute), track orthogonally; don't use additional polar tracking angles; and acquire object tracking points automatically. 1 Measures polar angles from selected objects (relative). 2 Uses polar tracking settings in object snap tracking. 4 Uses additional polar tracking angles (via PolarAng). 8 Acquire object snap tracking points by pressing Shift.
PolySides	4	Controls the default number of polygon sides; range is 3 to 1024.
Popups (R/O)	1	Reports display driver support of AUI: 0 Not available. 1 Available.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
PreviewEffect	2	Controls the visual effect for previewing selected objects: 0 Dashed lines. 1 Thick lines. 2 Thick dashed lines.
PreviewFilter	7	Controls the exclusion of objects from selection previewing (bitcode sum): 0 No objects excluded. 1 Objects on locked layers. 2 Objects in xrefs. 4 Tables. 7 Objects on locked layers, in xrefs, and tables. 8 Multiline text. 16 Hatch patterns. 32 Groups.
PreviewType	0	Specifies the view for drawing thumbnails: 0 Use last saved view. 1 Use Home view.
Product (R/O)	"AutoCAD"	Reports the name of the software.
Program (R/O)	"acad"	Reports the name of the software's executable file.
ProjectName	""	Controls the project name of the current drawing; searches for xref and image files.
ProjMode	1	Controls the projection mode for Trim and Extend commands: 0 Does not project. 1 Projects to xy plane of current UCS. 2 Projects to view plane.
ProxyGraphics	1	Toggles saving of proxy images in drawings: 0 Not saved; displays bounding box. 1 Image saved with drawing.
ProxyNotice	1	Toggles warning message displayed when drawing contains proxy objects.
ProxyShow	1	Controls the display of proxy objects: 0 Not displayed. 1 All displayed. 2 Bounding box displayed.
ProxyWebSearch	0	Toggles checking for object enablers: 0 Does not check for object enablers. 1 Checks for object enablers when an Internet connection is present.
PsLtScale	1	Toggles paper space linetype scaling: 0 Uses model space scale factor. 1 Uses viewport scale factor.
PsSolHeight	4.0	Specifies default height of polysolid objects.
PsSolWidth	0.25	Specifies default width of polysolid objects.
PsProlog	""	<i>Specifies the PostScript prologue file name.</i>

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
PsQuality	75	Controls resolution of PostScript display, in pixels: < 0 Display as outlines; no fill. 0 Displays no fills. > 0 Displays filled.
PStyleMode	1	Toggles the plot color matching mode of the drawing: 0 Uses named plot style tables. 1 Uses color-dependent plot style tables.
PStylePolicy (R/O)	1	Reports whether the object color is associated with its plot style: 0 Not associated. 1 Associated.
PsVpScale	0.0	Controls the view scale factor (ratio of units in paper space to units in newly-created model space viewports) 0 = scaled to fit.
PublishAllSheets	1	Determines which sheets (model space and layouts) are loaded automatically into the Publish command's list: 0 Current drawing only. 1 All open drawings.
PublishCollate	1	Controls how sheets are published: 0 Sheet sets are processed one sheet at a time; a separate plot is created for each sheet. 1 Sheets sets are processed as a single job and single plot file.
PublishHatch	1	Determines how hatch patterns are exported in DWF format to Impression: 0 Treated as separate objects. 1 Treated as a single object.
PUcsBase (R/O)	""	Reports name of UCS defining the origin and orientation of orthographic UCS settings; in paper space only.
Q		
QAFlags	0	Controls the quality assurance flags: 0 Turned off. 1 ^C metacharacter cancel grips, just as if user pressed Esc. 2 Long text screen listings do not pause. 4 Error and warning messages displayed in command line, instead of in dialog boxes. 128 Screen picks are accepted via the AutoLISP (command) function.
QcState	0	Toggles whether QuickCalc palette is open.
QpLocation	0	Locates the Quick Properties palette: 0 Near the cursor as defined by DSettings command's Quick Properties tab. 1 Floating, in a constant location.
QpMode	1	Controls Quick Properties palette display: 0 Off. 1 On for all objects 2 On only for objects defined CU.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
QTextMode	0	Toggles quick text mode.
QvDrawingPin	0	Toggles pinning of drawing quick views.
QvLayoutPin	0	Toggles pinning of layout quick views.
R		
R14RasterPlot	0	<i>Toggle; purpose unknown.</i>
RasterDpi	300	Controls the conversion of millimeters or inches to pixels, and vice versa; range is 100 to 32767.
RasterPercent	20	Percentage of system memory to allocate to plotting raster images; range is 0 - 100%.
RasterPreview (R/O)	1	Toggles creation of BMP preview image.
RasterThreshold	20	Specifies amount of RAM to allocate to plotting raster images; range is 0 - 2000MB.
RecoveryMode	2	Controls recording of drawing recovery information after software failure: 0 Note recorded. 1 Recorded; Drawing Recovery palette does not display automatically. 2 Recorded, and Drawing Recovery palette displays automatically.
RefEditName	""	Specifies the reference file name when in reference-editing mode.
RegenMode	1	Toggles regeneration mode: 0 Regens with each view change. 1 Regens only when required.
Re-Init	0	Controls the reinitialization of I/O devices: 1 Digitizer port. 2 <i>Plotter port; obsolete.</i> 4 Digitizer. 8 <i>Plotter; obsolete.</i> 16 Reloads PGP file.
RememberFolders	1	Toggles the path search method: 0 Path specified in AutoCAD desktop properties is default for file dialog boxes. 1 Last path specified by each file dialog box is remembered.
RenderPrefsState	1	Toggles display of the Render Preferences palette.
RenderUserLights	1	Determines which lights are rendered: 0 Default <i>or</i> user-defined lights are rendered. 1 Default <i>and</i> user-defined lights are rendered.
ReportError	1	Determines if AutoCAD sends an error report to Autodesk: 0 No error report created. 1 Error report is generated and sent to Autodesk.
RibbonContextSelect 	1	Determines when context-sensitive ribbon tabs are displayed: 0 Never. 1 When single-clicking objects or selection sets.. 2 When double-clicking objects or selection sets.

Variable Name	Default Value	Meaning
RibbonContextSelLim 	2500	Determines the maximum number of objects that activate context-sensitive ribbon tabs; range is 0 to 32767.
RibbonDockedHeight 	0	Determines the height of the ribbon when docked: 0 Ribbons sizes itself to height of the selected tab. 1 Minimum height in pixels. 500 Maximum height in pixels.
RibbonSelectionMode 	1	Determines fate of selections after command is selected from ribbon: 0 Pickfirst selection set is unselected. 1 Pickfirst selection set remains selected.
RibbonState (R/O)	1	Reports whether the ribbon is open.
RoamableRootPrefix (R/O)		Reports the path to the root folder where roamable customized files are located. "d:\documents and settings\username\application data\aut..."
RolloverTips	1	Toggles display of rollover tips.
RTDisplay	1	Toggles raster display during real-time zoom and pan: 0 Displays the entire raster image. 1 Displays raster outline only.
S		
SaveFidelity	1	Toggles how annotative objects are translated to earlier releases: 0 No changes made. 1 Each scale representation is saved to a separate layer.
SaveFile (R/O)	""	Reports the automatic save file name.
SaveFilePath	"...\temp\"	Specifies the path for automatic save files.
SaveName (R/O)	""	Reports the drawing's save-as file name.
SaveTime	10	Controls the automatic save interval, in minutes; 0 = disable auto save.
ScreenBoxes (R/O)	0	Reports the maximum number of menu items supported by display; 0 = screen menu turned off.
ScreenMode (R/O)	3	Reports the state of AutoCAD display: 0 Text screen. 1 Graphics screen. 2 Dual-screen display.
ScreenSize (R/O)	varies	Reports the current viewport size, in pixels, such as 719.0,381.0.
SDI	0	Controls the multiple-document interface (SDI is "single document interface"): 0 Turns on MDI. 1 Turns off MDI. (Only one drawing may be loaded into AutoCAD.) 2 Disables MDI for apps that cannot support MDI; read-only. 3 (R/O) Disables MDI for apps that cannot support MDI, even when SDI= 1.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
SelectionAnnoDisplay	1	Toggles how alternate scale representations are displayed when annotative objects selected: 0 Not displayed. 1 Displayed according to the value of XFadeCtl.
SelectionArea	1	Toggles use of colored selection areas.
SelectionAreaOpacity	25	Controls the opacity of color selection areas; range is 0 (transparent) to 100 (opaque).
SelectionPreview	3	Controls selection preview: 0 Off. 1 On when commands are inactive. 2 On when commands prompt for object selection.
SetBylayerMode	127	Controls which properties are affected by the SetByLayer command: 0 None. 1 Colors. 2 Linetypes. 4 Lineweights. 8 Materials. 16 Plot styles. 32 ByBlock is changed to Bylayer. 64 Blocks are changed from ByBlock to ByLayer. 127 All properties affected.
ShadEdge	3	Controls shading by Shade command: 0 Faces only shaded. 1 Faces shaded, edges in background color. 2 Only edges, in object color. 3 Faces in object color, edges in background color.
ShadeDif	70	Controls percentage of diffuse to ambient light; range is 0 to 100%.
ShadowPlaneLocation	0.0	Default height of the shadow plane.
ShortcutMenu	11	Controls display of shortcut menus (sum): 0 Does not display shortcut menus. 1 Displays default shortcut menus. 2 Displays edit shortcut menus. 4 Displays command shortcut menus when commands are active. 8 Displays command shortcut menus only when options available at command line. 16 Displays shortcut menus when the right button held down longer.
ShowHist	1	Toggles display of history in solids: 0 Does not display original solids. 1 Displays original solids depending on Show History property settings. 2 Displays all original solids.
ShowLayerUsage	1	Toggles layer-usage icons in Layers dialog box.
ShowMotionPin	1	Specifies pinning of Show Motion images.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
ShpName	""	Specifies the default shape name: Set to no default.
SigWarn	1	Toggles display of dialog box when drawings with digital signatures are opened: 0 Only when signature is invalid. 1 Always.
SketchInc	0.1	Controls the Sketch command's recording increment.
SkPoly	0	Toggles sketch line mode: 0 Record as lines. 1 Record as a polyline.
SkyStatus	0	Reports status of sun and sky background: 0 Sky is off. 1 Sky is on. 2 Sky and illumination are on.
SmoothMeshConvert 	0	Controls how solids and surfaces are converted to meshes: 0 Smooth models; optimizes or merges coplanar faces. 1 Smooth models; retains original mesh faces. 2 Flattened faces; optimizes or merges coplanar faces. 3 Flattened faces; retains original mesh faces.
SmoothMeshGrid 	3	Determines how the facet grid is displayed: 0 Not displayed. 1 Displayed for smoothing levels 0 and 1. 2 Displayed for smoothing levels 0, 1, and 2. 3 Displayed levels 0, 1, 2, and 3.
SmoothMeshMaxFace 	10000000	Specifies maximum faces on meshes; range is 1 to 16000000.
SmoothMeshMaxLev 	4	Specifies maximum smoothness level for meshes; range is 1 - 255.
SnapAng	0	Controls rotation angle for snap and grid; when not 0, grid lines are not displayed.
SnapBase	0.0,0.0	Controls current origin for snap and grid.
SnapIsoPair	0	Controls current isometric drawing plane: 0 Left isoplane. 1 Top isoplane. 2 Right isoplane.
SnapMode	0	Toggles snap mode.
SnapStyl	0	Toggles snap style: 0 Normal. 1 Isometric.
SnapType	0	Toggles snap for the current viewport: 0 Standard snap. 1 Polar snap.
SnapUnit	0.5,0.5	Controls x, y spacing for snap distances.
SolidCheck	1	Toggles solid validation.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
SolidHist	1	Toggles retention of history in solids.
SortEnts	127	Controls object display sort order: 0 Off. 1 Object selection. 2 Object snap. 4 Redraw. 8 Slide generation. 16 Regeneration. 32 Plot. 64 PostScript output.
SplFrame	0	Toggles polyline and mesh display: 0 Polyline control frame not displayed; display polygon fit mesh; 3D faces invisible edges not displayed. 1 Polyline control frame displayed; display polygon defining mesh; 3D faces invisible edges displayed.
SplineSegs	8	Controls number of line segments that define splined polylines; range is -32768 to 32767. <0 Drawn with fit-curve arcs. >0 Drawn with line segments.
SplineType	6	Controls type of spline curve: 5 Quadratic Bezier spline. 6 Cubic Bezier spline.
SsFound	""	Specifies path and file name of sheet sets.
SsLocate	1	Toggles whether sheet set files are opened with drawing: 0 Not opened. 1 Opened automatically.
SsmAutoOpen	1	Toggles whether the Sheet Set Manager is opened with drawing (SsLocate must be 1): 0 Not opened. 1 Opened automatically.
SsmPollTime	60	Controls time interval between automatic refreshes of status data in sheet sets; range is 20 to 600 seconds (SsmSheetStatus = 2).
SsmSheetStatus	2	Controls refresh of status data in sheet sets: 0 Does not refresh automatically. 1 Refreshes when sheet set is loaded or updated. 2 Also refreshes as specified by SsmPollTime.
SsmState (R/O)	0	Toggles whether Sheet Set Manager is open.
StandardsViolation	2	Controls whether alerts are displayed when CAD standards are violated: 0 Displays no alerts. 1 Displays alert when CAD standard violated. 2 Displays icon on status bar when file is opened with CAD standards, and when non-standard objects are created.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
Startup	0	Controls which dialog box is displayed by the New and QNew commands: 0 Displays Select Template dialog box. 1 Displays Startup and Create New Drawing dialog box.
StatusBar	1	Controls display of application and drawing status bars: 0 Hides both status bars. 1 Displays the application status bar. 2 Displays both status bars. 3 Displays the drawing status bar.
StepSize	6.0	Specifies length of steps in walk mode; range is 1E-6 to 1E+6.
StepsPerSec	2	Specifies speed of steps in walk mode; range is 1-30.
SubObjSelectionMode 	0	Specifies which subobjects are selected by Ctrl+click: 0 All. 1 Only vertices. 2 Only edges. 3 Only faces. 4 Only history subobjects.
SunPropertiesState	0	Toggles display of Sun Properties palette.
SunStatus	0	Toggles display of light by the sun.
SurfTab1	6	Controls density of m-direction surfaces and meshes; range 5 - 32766.
SurfTab2	6	Specifies density of n-direction surfaces and meshes; range 2 - 32766.
SurfType	6	Controls smoothing of surface by PEdit: 5 Quadratic Bezier spline. 6 Cubic Bezier spline. 8 Bezier surface.
SurfU	6	Controls surface density in m-direction; range is 2 to 200.
SurfV	6	Specifies surface density in n-direction; range is 2 to 200.
SysCodePage (R/O)	"ANSI_1252"	Reports the system code page; set by operating system.
T		
TableIndicator	1	Toggles display of column letters and row numbers during table editing.
TableToolbar	1	Toggles display of cell editing toolbar: 0 Does not display. 1 Displays. 2 Displays commands on ribbon.
TabMode	0	Toggles tablet mode.
Target (R/O)	0.0,0.0,0.0	Reports target coordinates in the current viewport.
Taskbar	1	Toggles whether each drawing appears as a button on the Windows taskbar.
TbCustomize	1	Toggles whether toolbars can be customized.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
TDCreate (R/O)	<i>varies</i>	Reports the date and time that the drawing was created, such as 2448860.54014699.
TDInDwg (R/O)	<i>varies</i>	Reports the duration since the drawing was loaded, such as 0.00040625.
TDuCreate (R/O)	<i>varies</i>	Reports the universal date and time when the drawing was created, such as 2451318.67772165.
TDUpdate (R/O)	<i>varies</i>	Reports the date and time of last update, such as 2448860.54014699.
TDUsrTimer (R/O)	<i>varies</i>	Reports the decimal time elapsed by user-timer, such as 0.00040694.
TDuUpdate (R/O)	<i>varies</i>	Reports the universal date and time of the last save, such as 2451318.67772165.
TempOverrides	1	Toggles temporary overrides.
TempPrefix (R/O)	"d:\temp"	Reports the path for temporary files set by Temp variable.
TextEditor (R/O) 	0	Reports the status of the text editor: 0 Closed. 1 Open.
TextEval	0	Toggles the interpretation of text input during the -Text command: 0 Literal text. 1 Read (and ! as AutoLISP code.
TextFill	1	Toggles the fill of TrueType fonts when plotted: 0 Outline text. 1 Filled text.
TextOutputFileFormat	0	Controls Unicodes for plot and text window log files: 0 ANSI format. 1 UTF-8 (Unicode). 2 UTF-16LE (Unicode). 3 UTF-16BE (Unicode).
TextQlty	50	Controls the resolution of TrueType fonts when plotted; range is 0 to 100.
TextSize	0.2	Controls the default height of text (2.5 in metric units).
TextStyle	"Standard"	Specifies the default name of text style.
Thickness	0.0	Controls the default object thickness.
ThumbSize	1	Specifies the size of thumbnail images: 0 64x64 pixels. 1 128x128 pixels. 2 256x256 pixels.
TileMode	1	Toggles the view mode: 0 Displays layout tab. 1 Displays model tab.
TimeZone	-80000	Specifies current time zone.
ToolTipMerge	0	Toggles the merging of tooltips during dynamic display.
ToolTips	1	Toggles the display of tooltips.
TpState (R/O)	0	Reports if Tool Palettes palette is open.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
TraceWid	0.0500	Specifies current width of traces.
TrackPath	0	Controls display of polar and object snap tracking alignment paths: 0 Displays object snap tracking path across the entire viewport. 1 Displays object snap tracking path between the alignment point and “From point” to cursor location. 2 Turns off polar tracking path. 3 Turns off polar and object snap tracking paths.
TrayIcons	1	Toggles the display of the tray on status bar.
TrayNotify	1	Toggles service notifications displayed by the tray.
TrayTimeout	0	Controls length of time that tray notifications are displayed; range is 0 to 10 seconds.
TreeDepth	3020	Controls the maximum branch depth (in <i>xyy</i> format): <i>xx</i> Model-space nodes. <i>yy</i> Paper-space nodes. <i>>0</i> 3D drawing. <i><0</i> 2D drawing.
TreeMax	10000000	Controls the memory consumption during drawing regeneration.
TrimMode	1	Toggles trims during Chamfer and Fillet: 0 Leaves selected edges in place. 1 Trims selected edges.
TSpaceFac	1.0	Controls the mtext line spacing distance measured as a factor of “normal” text spacing; ranges from 0.25 to 4.0.
TSpaceType	1	Controls the type of mtext line spacing: 1 At Least: adjusts line spacing based on the height of the tallest character in a line of mtext. 2 Exactly: uses the specified line spacing; ignores character height.
TStackAlign	1	Controls vertical alignment of stacked text: 0 Bottom aligned. 1 Center aligned. 2 Top aligned.
TStackSize	70	Controls size of stacked text as a percentage of the current text height; range is 25 to 125%.
U		
UcsAxisAng	90	Controls the default angle for rotating the UCS around an axis (via the UCS command using the X, Y, or Z options); valid values limited to: 5, 10, 15, 18, 22.5, 30, 45, 90, or 180.
UcsBase	""	Specifies name of UCS that defines the origin and orientation of orthographic UCS settings.
UcsDetect	1	Toggles dynamic UCS mode.
UcsFollow	0	Toggles view displayed with new UCSs: 0 No change. 1 Automatically aligns UCS with new view.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
 UcsIcon	3	Controls display of the UCS icon: 0 Off. 1 On at lower-left corner. 2 On at UCS origin, if possible.
UcsName (R/O)	""	Reports the name of current UCS view: "" Current UCS is unnamed.
UcsOrg (R/O)	0.0,0.0,0.0	Reports origin of current UCS relative to WCS.
UcsOrtho	1	Controls whether the related orthographic UCS settings are restored automatically: 0 UCS setting remains unchanged when orthographic view is restored. 1 Related ortho UCS is restored automatically when an ortho view is restored.
UcsView	1	Toggles whether the current UCS is saved with a named view.
UcsVp	1	Toggles whether the UCS in active viewports remains fixed (locked) or changes (unlocked) to match the UCS of the current viewport.
UcsXDir (R/O)	1.0,0.0,0.0	Reports the x-direction of current UCS relative to WCS.
UcsYDir (R/O)	0.0,1.0,0.0	Reports the y-direction of current UCS relative to WCS.
UndoCtl (R/O)	21	Reports the status of undo (bitsum): 0 Disabled. 1 Enabled. 2 Limited to one command. 4 Set to auto-group mode. 8 Active group mode. 16 Combined zooms and pans.
UndoMarks (R/O)	0	Reports the number of undo marks.
UnitMode	0	Toggles the type of units display: 0 As set by Units command. 1 As entered by user.
UOSnap 	1	Toggles object snapping for geometry in DWF, DWFx, PDF, and DGN underlays; overrides the DwfOsnap, PdfOsnap, and DgnOsnap system variables: 0 Disabled. 1 Enabled. 2 Setting dependent on DwfOsnap, PdfOsnap, and DgnOsnap.
UpdateThumbnail	15	Controls how thumbnails are updated (sum): 0 Thumbnail previews not updated. 1 Sheet views updated. 2 Model views updated. 4 Sheets updated. 8 Updated when sheets or views are created, modified, or restored. 16 Updated when the drawing is saved.
UserI1 thru UserI5	0	Stores user-definable integer variables.

Variable Name	Default Value	Meaning
UserR1 thru UserR5	0.0	Stores user-definable real variables.
UserS1 thru UserS5	""	Stores user-definable string variables; values are not saved.
V		
ViewCtr (R/O)	<i>varies</i>	Reports x, y, z coordinates of center of current view, such as 15,9,56.
ViewDir (R/O)	<i>varies</i>	Reports current view direction relative to UCS: 0,0,1=plan view.
ViewMode (R/O)	0	Reports the current view mode: 0 Normal view. 1 Perspective mode on. 2 Front clipping on. 4 Back clipping on. 8 UCS-follow on. 16 Front clip not at eye.
ViewSize (R/O)	<i>varies</i>	Reports the height of current view in drawing units.
ViewTwist (R/O)	0	Reports the twist angle of current view.
VisRetain	1	Controls xref drawing's layer settings: 0 Xref-dependent layer settings are not saved in the current drawing. 1 Xref-dependent layer settings are saved in the current drawing, and take precedence over settings in the xref'ed drawing the next time the current drawing is loaded.
VpLayerOverrides (R/O)	0	Reports whether VP layer properties are overridden in the current viewport: 0 None. 1 At least one layer.
VpLayerOverridesMode	1	Controls display and plot of VP layer property overrides: 0 Not displayed or plotted. 1 Displayed and plotted.
VpMaximizedState (R/O)	0	Reports whether viewport is maximized by VpMax command.
VpRotateAssoc 	1	Toggles whether the contents are rotated with viewports: 0 Views are not rotated by viewports. 1 Views are rotated by viewports.
VsBackgrounds	1	Toggles display of backgrounds in visual styles.
VsEdgeColor	"ByEntity"	Specifies the edge color.
VsEdgeJitter	-2	Specifies the level of pencil effect: 0 or -n None. 1 Low. 2 Medium. 3 High.
VsEdgeOverhang	-6	Specifies extension of pencil lines beyond edges; range is 1 to 100 pixels; -n = none.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
VsEdges	1	Specifies types of edges to display: 0 No edges displayed. 1 Isolines displayed. 2 Facets and edges displayed.
VsEdgeSmooth	1	Specifies crease angle; range is 0 to 180 degrees.
VsFaceColorMode	0	Determines color of faces.
VsFaceHighlight	-30	Specifies the size of highlights; range is -100 to 100. Ignored when VsMaterialMode is 1 or 2 and objects have materials attached.
VsFaceOpacity	-60	Controls the transparency/opacity of faces; range is -100 to 100 (fully opaque).
VsFaceStyle	0	Determines how faces are displayed: 0 No rendering. 1 Real. 2 Gooch.
VsHaloGap	0	Specifies halo gap; range is 0 to 100 pixels.
VsHidePrecision	0	Toggles accuracy of hides and shades.
VsIntersectionColor	“7 (white)”	Specifies the color of intersecting polylines.
VsIntersectionEdges	0	Toggles the display of intersecting edges.
VsIntersectionLtype	1	Specifies the linetype for intersecting lines: 0 Off. 1 Solid. 2 Dashed. 3 Dotted. 4 Short dash. 5 Medium dash. 6 Long dash. 7 Double-short dash. 8 Double-medium dash. 9 Double-long dash. 10 Medium-long dash. 11 Sparse dot.
VsIsoOntop	0	Toggles whether isolines are displayed.
VsLightingQuality	1	Toggles the quality of lighting: 0 Displays facets. 1 Smooths facets. 2 Turns on per-pixel lighting.
VsMaterialMode	0	Controls the display of material finishes: 0 No materials displayed. 1 Materials displayed. 2 Materials and textures displayed.
VSMax (R/O)	<i>varies</i>	Reports the upper-right corner of virtual screen, such as 37.46,27.00,0.00.

<i>Variable Name</i>	<i>Default Value</i>	<i>Meaning</i>
VSMIn (R/O)	<i>varies</i>	Reports the lower-left corner of virtual screen, such as -24.97,-18.00,0.0.
VsMonoColor	"RGB:255,255,255"	Specifies the monochrome tint.
VsObscuredColor	"ByEntity"	Specifies the color of obscured lines.
VsObscuredEdges	1	Toggles the display of obscured edges.
VsObscuredLtype	1	Specifies the linetype of obscured lines; see VsIntersectionLtype.
VsShadows	0	Determines the quality of shadows: 0 No shadows displayed. 1 Ground shadows displayed. 2 Full shadows displayed.
VsSilhEdges	0	Toggles the display of silhouette edges.
VsSilhWidth	5	Specifies the width of silhouette edge lines; range is 1 to 25 pixels.
VsState	0	Toggles the Visual Styles palette.
VtDuration	750	Controls the duration of smooth view transition; range is 0 to 5000 seconds.
VtEnable	3	Controls smooth view transitions for pans, zooms, view rotations, and scripts: 0 Turned off. 1 Pan and zoom. 2 View rotation. 3 Pan, zoom, and view rotation. 4 During scripts only. 5 Pan and zoom during scripts. 6 View rotation during scripts. 7 Pan, zoom, and view rotation during scripts.
VtFps	7	Controls minimum speed for smooth view transitions; range is 1 to 30 frames per second.
W		
WhipArc	0	Toggles display of circular objects: 0 Displays as connected vectors. 1 Displays as true circles and arcs.
WhipThread	1	Controls multithreaded processing on two CPUs (if present) during redraws and regens: 0 Performs single-threaded calculations. 1 Regenerations multi-threaded. 2 Redraws multi-threaded. 3 Regens and redraws multi-threaded.
WindowAreaColor	150	Specifies ACI color of windowed selection area.
WmfBkgnd	Off	Toggles background of .wmf files: Off Background is transparent. On Background is same as AutoCAD's background color.

Variable Name	Default Value	Meaning
WmfForegnd	Off	Toggles foreground colors of exported WMF images: Off Foreground is darker than background. On Foreground is lighter than background.
WorldUcs (R/O)	1	Toggles matching of WCS with UCS: 0 Current UCS does not match WCS. 1 UCS matches WCS.
WorldView	1	Toggles view during 3dOrbit, DView, and VPoint commands: 0 Current UCS. 1 WCS.
WriteStat (R/O)	1	Toggle: reports whether <i>.dwg</i> file is read-only: 0 Drawing file cannot be written to. 1 Drawing file can be written to.
WsCurrent	"varies"	Controls name of current workspace.
X		
XClipFrame	2	Toggles visibility of xref clipping boundary: 0 Hides and does not plot the xclip frame. 1 Displays and plots xclip frames. 2 Displays but does not plot xclip frames.
XDwgFadeCtl 	70	Specifies the amount attached xrefs are shown faded: 0 Not faded. >0Increasingly faded, to a maximum of 90%. <0Not faded, but fade value is retained.
XEdit	1	Toggles editing of xrefs: 0 Cannot in-place refedit. 1 Can in-place refedit.
XFadeCtl	50	Controls faded display of objects not being edited in-place: 0 No fading; minimum value. 90 90% fading; maximum value.
XLoadCtl	2	Controls demand loading: 0 Demand loading turned off; entire drawing is loaded. 1 Demand loading turned on; xref file opened. 2 Demand loading turned on; a <i>copy</i> of the xref file is opened.
XLoadPath	"...\temp"	Specifies path for storing temporary copies of demand-loaded xref files.
XRefCtl	0	Toggles creation of <i>.xlg</i> xref log files.
XrefNotify	2	Controls notification of updated and missing xrefs: 0 Displays no alert. 1 Displays icon indicating xrefs are attached; yellow alert indicates missing xrefs. 2 Also displays balloon messages when an xref is modified.

Variable Name	Default Value	Meaning
XrefType	0	Toggles xrefs: 0 Attached. 1 Overlaid.
Z		
ZoomFactor	60	Controls the zoom level via mouse wheel; range from 3 to 100.
ZoomWheel	0	Switches the zoom direction when mouse wheel is rotated forward: 0 Zooms in (opposite of most other software, including Inventor). 1 Zooms out.
3		
3dConversionMode	1	Controls how material and light definitions are converted when pre-AutoCAD 2008 drawings are opened: 0 None converted. 1 Converted automatically. 2 Converted after prompted.
3dDwfPrc	2	Level of precision in drawings exported as .dwf files: 1 1 2 0.5 3 0.2 4 0.1 5 0.01 6 0.001
3dSelectionMode	1	Toggles how visually overlapping objects are selected (other than in 2D and 3D wireframe visual modes; this system variable is slated for removal in a future release of AutoCAD): 0 Uses traditional 3D selection. 1 Uses line-of-sight selection.

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Summary of Dimension Variables

The following sections summarize the meaning of dimension variables, grouped by purpose. (For more on dimension variables, see chapter 12 on editing dimensions.)

General

DimAnno (annotative) reports whether the current dimension style uses annotative scaling; read-only. When on (set to 1), **DIMSCALE** is set to 0.

DimAso (associative) toggles dimensions between associative and non-associative.

DimAssoc (associative) determines how dimensions are created: exploded, attached to defpoints, or attached to objects.

DimScale specifies the overall scale factor for dimensions. When set to zero, AutoCAD calculates the scale factor for dimensions by two methods:

- When **DIMANNO** = 1, AutoCAD divides the scale factor of the current model space viewport by that of paper space.
- When **DIMANNO** = 0, AutoCAD uses the current viewport scale factor.

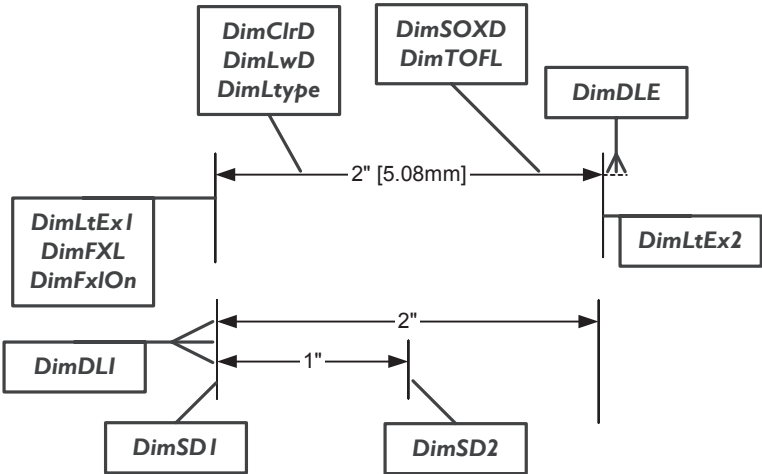
DimSho (show) updates dimensions while dragging.

DimStyle specifies the current dimension style as set by the **DIMSTYLE** command. (Note that the system variable and the command share the same name, so use the **SETVAR** command to access the dimvar.)

NEW TO AUTOCAD 2010 IN THIS APPENDIX

- **DIMTXTDIRECTION** system variable is new; **DIMTAD** has an extra option (4).
- **CCONSTRAINTFORM**, **CDYNDISPLAYMODE**, **CONSTRAINTNAMEFORMAT**, **DIMCONSTRAINTICON**, and **DYNCONSTRAINTDISPLAY** system variables are added in support of dimensional constraints; see Chapter 13, “Applying Dimension Constraints.”

Dimension Lines



Extension and Offset

DimDLE (dimension line extension) specifies how far the dimension line extends past the extension lines.

DimLwD (lineweight dimension) specifies the lineweight of dimension lines.

DimDLI (dimension line increment) specifies the offset for baseline dimensions.

Suppression and Position of Dimension Lines

DimSD1 (suppress dimension 1) suppresses the first dimension line.

DimSD2 (suppress dimension 2) suppresses the second dimension line.

DimSOXD (suppress outside extension dimension) suppresses dimension lines outside extension lines.

DimTOFL (text outside forced line) forces dimension lines inside extension lines when text is outside the extension lines.

Color and Lineweight

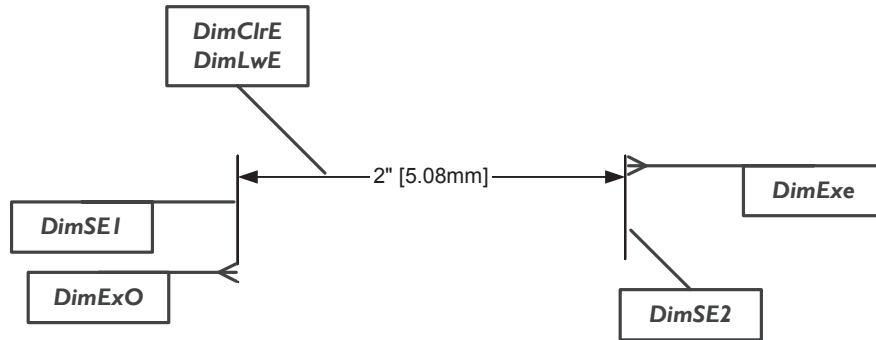
DimClrD (color dimension) specifies the color of dimension lines.

DimLwD (lineweight dimension) specifies the lineweight for dimension lines.

Linetype

DimLtype (linetype) specifies the linetype for dimension lines.

Extension Lines



Suppression and Linetype

DimFXL (fixed-length extension lines) specifies the default length of fixed-length extension lines.

DimFxlOn (fixed-length extension lines on) toggles the use of fixed-length extension lines.

DimLtEx1 (linetype for extension line 1) specifies the linetype for the first extension line.

DimLtEx2 (linetype for extension line 2) specifies the linetype for the second extension line.

Color and Lineweight

DimClrE (color extension) specifies the color of extension lines.

DimLwE (lineweight extension) specifies the lineweight of extension lines.

Extension and Offset

DimExE (extension extend) extends the extension lines above the dimension line.

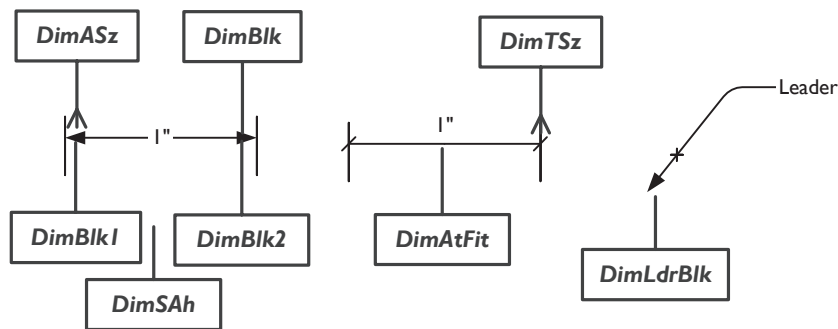
DimExO (extension offset) offsets the extension line from the origin.

Suppression and Position

DimSE1 (suppress extension 1) suppresses the first extension line.

DimSE2 (suppress extension 2) suppresses the second extension line.

Arrowheads



Size and Fit

DimASz (arrow size) determines the length of arrowhead blocks.

DimTSz (tick size) specifies the size of tick strokes.

DimATFit (arrow text fit) determines under which conditions arrowheads and text are fitted between extension lines; see Alphabetical Listing of Dimvars later in this chapter.

Names of Blocks

DimBlk (block) names the arrowhead block.

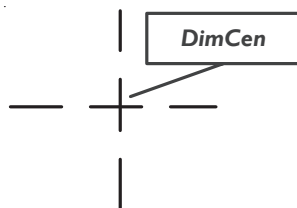
DimBlk1 (block 1) names the first arrowhead block.

DimBlk2 (block 2) names the second arrowhead block.

DimLdrBlk (leader block) names the leader arrowhead.

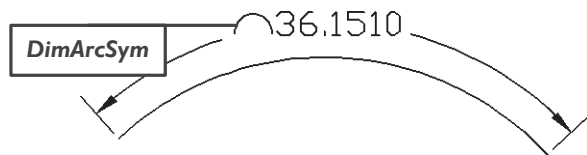
DimSAh (separate arrowheads) determines whether separate arrowhead blocks are used.

Center Marks



DimCen (center) specifies the mark size and line.

Arc Length Dimensions



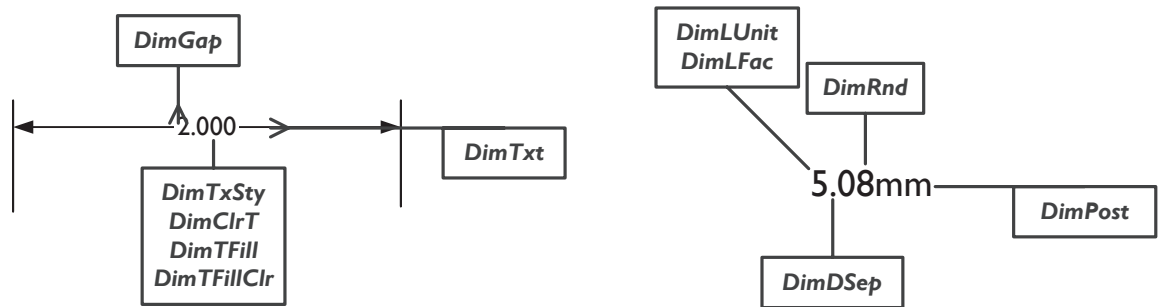
DimArcSym (arc symbol) specifies the location of the arc length symbol.

Jogged Dimensions



DimJogAng (jog angle) specifies the default angle of jogged radial dimension lines.

Text



Color and Format of Text

DimClrT (color text) specifies the color of the text.

DimGap determines the gap between the dimension line and text.

DimTxSty (text style) specifies the text style.

DimTxt (text) stores the text height.

 **DimTxtDirection** specifies the reading direction of vertical and horizontal text.

Background Fill

DimTFill (text fill) switches the background fill of dimension text among these: none, same as drawing background color, or as specified by DIMTFILLCLR.

DimTFillClr (text fill color) specifies the color of the background fill.

Units, Scale, and Precision

- DimLUnit** (linear unit) specifies the format of linear units.
- DimPost** (postfix) specifies prefixes and suffixes of dimension text.
- DimLFac** (linear factor) specifies the linear unit scale factor.
- DimDSep** (decimal separator) specifies the decimal separator.
- DimRnd** (round) rounds distances.
- DimFrac** (fractions) determines how fractions are stacked: horizontal, diagonal, or not stacked.



- DimZIn** (zero inches) suppresses zeroes in feet-inches units.
- DimADec** (angular decimal) specifies the precision of angular dimensions.
- DimAUnit** (angular unit) specifies the format for angular dimensions.
- DimAZIn** (angular zero inches) controls how zeros are suppressed in angular dimensions.

Justification

- DimJust** (justify) specifies alignment (justification) of horizontal text: left, center, right.
- DimTAD** (text adjust dimension) positions dimension text vertically: top, center, outside, JIS.
- DimTVP** (text vertical position) specifies the vertical text position when DIMITAD is off.
- DimTIH** (text inside horizontal) determines whether text inside extensions is horizontal.
- DimTIX** (text in extension) forces text between extension lines.
- DimTMove** (text move) determines how dimension text is relocated: with dimension line, an added leader, or without a leader.
- DimTOH** (text outside horizontal) determines whether text outside extension lines is horizontal.
- DimUPT** (user position text) toggles whether user positions dimension line and/or text.

Alternate Text

DimAlt (alternate) toggles alternate units.

DimAltU (alternate units) determines the format of alternate units.

DimAltD (alternate decimals) specifies decimal places of alternate units.

DimAltF (alternate factor) specifies the scale factor of alternate units.

DimAltRnd (alternate rounding) rounds off alternate units.

DimAltZ (alternate zero) suppresses zeros in alternate units.

DimAPost (alternate postfix) determines the prefixes and suffixes for alternate text.

Limits

DimLim (limits) toggles dimension limits.

Tolerance Text

DimTol (tolerance) toggles whether tolerances are drawn.

DimTZIn (tolerance zero inches) suppresses zeros in tolerances.

DimTFac (tolerance factor) scales the tolerance text height.

DimTolJ (tolerance justify) justifies tolerance text vertically.

DimTP (tolerance plus) specifies the plus tolerance value.

DimTM (tolerance minus) specifies the minus tolerance value.

Primary Tolerance

DimDec (decimals) specifies the decimal places for the primary tolerance.

DimTDec (tolerance decimals) specifies the decimal places for primary tolerance units.

Alternate Tolerance

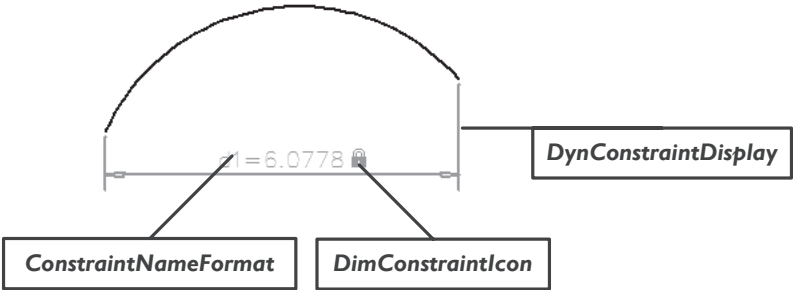
DimAltTD (alternate tolerance decimals) specifies decimal places for tolerance alternate units.

DimAltTZ (alternate tolerance zero) suppresses zeros in tolerance alternate units.

 DIMENSIONAL CONSTRAINTS

CConstraintForm (current constraint form) controls whether annotational or dynamic constraints are applied to objects.

ConstraintNameFormat specifies the type of text displayed by dimensional constraints: variable names, values, or variable names and expressions.



DimConstraintIcon toggles the lock icon next to dimensional constraint text.

DynConstraintDisplay displays and hides dynamic constraints.

ALPHABETICAL LISTING OF DIMVARS

The table summarizes dimension variables, default values in the *acad.dwt* template, and optional values.

DimVar	Default	Settings and Options
A		
DimADec	0	Specifies angular dimension precision: -1 Use DimDec setting (default). 0 Zero decimal places (minimum). 8 Eight decimal places (maximum).
DimAlt	Off	Specifies alternate units: On Enabled. Off Disabled.
DimAltD	2	Specifies alternate unit decimal places.
DimAltF	25.4000	Specifies alternate unit scale factor.
DimAltRnd	0.0000	Specifies rounding factor of alternate units.
DimAltTD	2	Specifies tolerance alternate unit decimal places.
DimAltTZ	0	Specifies how alternate tolerance units zeros are handled: 0 Zeros not suppressed. 1 Suppresses all zeros. 2 Includes 0 feet, but suppresses 0 inches. 3 Includes 0 inches, but suppresses 0 feet. 4 Suppresses leading zeros. 8 Suppresses trailing zeros.
DimAltU	2	Specifies format of alternate units: 1 Scientific. 2 Decimal. 3 Engineering. 4 Architectural; stacked. 5 Fractional; stacked. 6 Architectural. 7 Fractional. 8 Windows desktop units setting.
DimAltZ	0	Specifies zero suppression for alternate units: 0 Suppresses 0 ft and 0 in. 1 Includes 0 ft and 0 in. 2 Includes 0 ft; suppress 0 in. 3 Suppresses 0 ft; include 0 in. 4 Suppresses leading 0 in decimal dims. 8 Suppresses trailing 0 in decimal dims. 12 Suppresses leading and trailing zeroes.
DimAnno	0	Reports whether the current dimension style is annotative.
DimAPost	""	Specifies prefix and suffix for alternate text.

DimVar	Default	Settings and Options
DimArcSym	""	Specifies position of the arc length symbol: 0 Before the dimension text. 1 Above dimension text. 2 Not displayed.
DimAso	On	Toggles associative dimensions: On Dimensions are associative. Off Dimensions are not associative.
DimAssoc	2	Controls creation of dimensions: 0 Dimension elements are exploded. 1 Single dimension object, attached to defpoints. 2 Single dimension object, attached to geometric objects.
DimASz	0.1800	Specifies arrowhead length.
DimAtFit	3	Specifies how to handle text and arrows when space is insufficient between extension lines: 0 Text and arrows outside extension lines. 1 Arrows first outside, then text. 2 Text first outside, then arrows. 3 Either text or arrows, whichever fits better.
DimAUnit	0	Specifies the angular dimension format: 0 Decimal degrees. 1 Degrees.Minutes.Seconds. 2 Grad. 3 Radian. 4 Surveyor units.
DimAZin	0	Suppresses zeros in angular dimensions: 0 Display all leading and trailing zeros. 1 Suppress 0 in front of decimal. 2 Suppress trailing zeros behind decimal. 3 Suppress zeros in front and behind the decimal.

B

DimBlk	""	Specifies arrowhead block name: Architectural tick: "Archtick" Box filled: "Boxfilled" Box: "Boxblank" Closed blank: "Closedblank" Closed filled: "" (default) Closed: "Closed" Datum triangle filled: "Datumfilled" Datum triangle: "Datumblank" Dot blanked: "Dotblank" Dot small: "Dotsmall" Dot: "Dot" Integral: "Integral"
--------	----	--

DimVar	Default	Settings and Options
		None: "None" Oblique: "Oblique" Open 30: "Open30" Open: "Open" Origin indication: "Origin" Rightangle: "Open90"
DimBlk1	""	Names the first arrowhead's block; uses same names as DimBlk. . Return to default arrowhead.
DimBlk2	""	Names the second arrowhead's block. . Return to default arrowhead.

C

DimCen	0.0900	Specifies the center mark size: - <i>n</i> Draws center lines. 0 No center mark or lines drawn. + <i>n</i> Draws center marks of length <i>n</i> .
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D

DimClrD	0	Specifies the dimension line color: 0 BYBLOCK (default). 1 Red. ... 255 Dark gray. 256 BYLAYER.
DimClrE	0	Specifies the extension line and leader color.
DimClrT	0	Specifies the dimension text color.
DimDec	4	Specifies the number of primary tolerance decimal places.
DimDLE	0.0000	Specifies the length of the dimension line extension.
DimDLI	0.3800	Specifies the length of the dimension line continuation increment.
DimDSep	","	Specifies the decimal separator (must be a single character).

E

DimExe	0.1800	Specifies the length of extension above dimension line.
DimExO	0.0625	Specifies the extension line origin offset.

F

DimFrac	0	Fraction format when DimLUnit is set to 4 or 5: 0 Horizontal. 1 Diagonal. 2 Not stacked.
DimFXL	1.0000	Specifies the default length of fixed-length extension lines.
DimFxlOn	Off	Toggles fixed-length extension lines: On Extension lines are displayed at a fixed length. Off Extension lines are displayed at the user-drawn length

DimVar	Default	Settings and Options
G		
DimGap	0.0900	Specifies the gap between the dimension line and the text.
J		
DimJogAng	45	Specifies the default angle for jogged radial dimension lines.
DimJust	0	Specifies the horizontal text positioning: 0 Center justify. 1 Next to first extension line. 2 Next to second extension line. 3 Above first extension line. 4 Above second extension line.
L		
DimLdrBlk	""	Specifies block name for leader arrowhead; same names as DimBlock . . Return to default.
DimLFac	1.0000	Specifies the linear unit scale factor.
DimLim	Off	Toggles generation of dimension limits.
DimLtEx1	""	Specifies the linetype used for the first extension line.
DimLtEx2	""	Specifies the linetype used for the first extension line.
DimLtype	""	Specifies the linetype used for dimension lines.
DimLUnit	2	Specifies the linear dimension units (except angular): 1 Scientific. 2 Decimal. 3 Engineering. 4 Architectural. 5 Fractional. 6 Windows desktop.
DimLwD	-2	Specifies the dimension line lineweight; valid values are BYLAYER, BYBLOCK, or an integer multiple of 0.01 mm.
DimLwE	-2	Specifies the extension lineweight; valid values are BYLAYER, BYBLOCK, or an integer multiple of 0.01 mm.
P		
DimPost	""	Specifies the default prefix or suffix for dimension text (maximum 13 characters): " No suffix. <>mm Millimeter suffix. <>Å Angstrom suffix.
R		
DimRnd	0.0000	Specifies the rounding value for dimension distances.

DimVar	Default	Settings and Options
S		
DimSAh	Off	Specifies whether separate arrowhead blocks are used: Off Use arrowhead defined by DimBlk. On Use arrowheads defined by DimBlk1 and DimBlk2.
DimScale	1.0000	Specifies the overall scale factor for dimensions: 0 Value is computed from the scale between current model space viewport and paper space. >0 Scales text and arrowheads.
DimSD1	Off	Toggles suppression of the first dimension line: On First dimension line is suppressed. Off Not suppressed.
DimSD2	Off	Toggles suppression of the second dimension line: On Second dimension line is suppressed. Off Not suppressed.
DimSE1	Off	Toggles suppression of the first extension line: On First extension line is suppressed. Off Not suppressed.
DimSE2	Off	Toggles suppression of the second extension line: On Second extension line is suppressed. Off Not suppressed.
DimSho	On	Toggles the updating of dimensions while dragging: On Dimensions are updated during drag. Off Dimensions are updated after drag.
DimSOXD	Off	Toggles suppression of dimension lines outside extension lines: On Dimension lines not drawn outside extension lines. Off Are drawn outside extension lines.
DimStyle	"STANDARD"	Name of the current dimension style.
T		
DimTAD	0	Vertical position of dimension text: 0 Centered between extension lines. 1 Above dimension line, except when dimension line not horizontal and DimTIH = 1. 2 On side of dimension line farthest from the defining points. 3 Conforms to JIS. 4 Places text below dimension line  .
DimTDec	4	Specifies the number of primary tolerance decimal places.
DimTFac	1.0000	Specifies the tolerance text height scaling factor.

<i>DimVar</i>	<i>Default</i>	<i>Settings and Options</i>
DimTFill	0	Toggles the display of background fill: 0 None. 1 Background color of drawing. 2 Color specified by DimTFillClr.
DimTFillClr	0	Specifies the color used for the background of dimension text.
DimTIH	On	Toggles the alignment of text inside extensions: Off Text aligned with dimension line. On Text is horizontal.
DimTIX	Off	Toggles the placement of text inside extensions: Off Place text inside extension lines, if room. On Force text between the extension lines.
DimTM	0.0000	Specifies the minus tolerance.
DimTMove	0	Determines how dimension text is moved: 0 Dimension line moves with text. 1 Adds a leader when text is moved. 2 Text moves anywhere; no leader.
DimTOFL	Off	Forces line inside extension lines: Off Dimension lines not drawn when arrowheads are outside. On Dimension lines drawn, even when arrowheads are outside.
DimTOH	On	Toggles the alignment of text outside extension lines: Off Text aligned with dimension line. On Text is horizontal.
DimTol	Off	Generate dimension tolerances: Off Tolerances not drawn. On Tolerances are drawn.
DimTolJ	1	Specifies the vertical justification of tolerance text: 0 Bottom. 1 Middle. 2 Top.
DimTP	0.0000	Specifies the plus tolerance value.
DimTSz	0.0000	Specifies the size of oblique tick strokes: 0 Arrowheads. >0 Oblique strokes.
DimTVP	0.0000	Specifies the text vertical position when DimTAD=0 : 1 Turns on DimTAD . >-0.7 or <0.7 Dimension line is split for text.
DimTxSty	"STANDARD"	Specifies the dimension text style.
DimTxt	0.1800	Specifies the text height.

DimVar	Default	Settings and Options
 DimTxtDirection		
	0	Specifies the reading direction of text: 0 Horizontal text reads left to right; Vertical text reads bottom to top. 1 Horizontal text read right to left; Vertical text reads top to bottm.
DimTZin	0	Specifies tolerance zero suppression: 0 Suppress 0 ft and 0 in. 1 Include 0 ft and 0 in. 2 Include 0 ft; suppress 0 in. 3 Suppress 0 ft; include 0 in. 4 Suppress leading 0 in decimal dim. 8 Suppress trailing 0 in decimal dim. 12 Suppress leading and trailing zeroes.

U

DimUPT	Off	User-positioned text: Off Cursor positions dimension line. On Cursor also positions text.
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Z

DimZIN	0	Specifies the suppression of 0 in feet-inches units: 0 Suppress 0 ft and 0 in. 1 Include 0 ft and 0 in. 2 Include 0 ft; suppress 0 in. 3 Suppress 0 ft; include 0 in. 4 Suppress leading 0 in decimal dim. 8 Suppress trailing 0 in decimal dim. 12 Suppress leading and trailing zeroes.
---------------	---	---

Two dimvars are considered obsolete by Autodesk, but remain in AutoCAD for compatibility reasons:

Obsolete DimVar	Default	Comments
DimFit	3	Uses DimATfit and DimTMove instead.
DimUnit	2	Replaced by DimLUnit and DimFrac.

 The following system variables are used by dimensional constraints:

Sysar	Default	Comments
CConstraintForm		
	0	Controls whether annotational or dynamic constraints are applied: 0 Applies dynamic constraints. 1 Applies annotative constraints.
ConstraintNameFormat		
	3	Specifies the type of text displayed by dimensional constraints: 0 Variable names. 1 Values only. 2 Variable names and expressions.
DimConstraintIcon		
	3	Toggles the lock icon next to dimensional constraint text: 0 Does not display the lock icon. 1 Displays icon next to dynamic constraints only. 2 Displays the icon next to annotational constraints only. 3 Displays the icon next to dynamic and annotational constraints.
DynConstraintDisplay		
	1	Displays and hides dynamic constraints: 0 Off. 1 On.

AutoCAD Fonts, Linetypes, Hatch Patterns, Gradients & Lineweights

Fonts

The fonts shown below are just some of the many provided with AutoCAD in SHX and TrueType format. SHX fonts are based on Autodesk's design, while TrueType are based on Microsoft technology. In addition to the fonts provided by Autodesk, AutoCAD can use any TrueType font found on your computer. SHX fonts are stored in the `\autocad 2010\fonts` folder, while TTF fonts are stored in the `\windows\fonts` folder. The default font used by the default style is *arial.ttf*.

Complex

ABCDEFGHIJKLMNOPQRSTUVWXYZ
abcdefghijklmnopqrstuvwxyz
0123456789 ° ± ø
!@#%\$%^&*()_+-=,./<>?:':"[]{}|\'~

Country Blueprint

ABCDEFGHIJKLMNOPQRSTUVWXYZ
abcdefghijklmnopqrstuvwxyz
0123456789 ° ± ø
!@#%\$%^&*()_+-=,./<>?:':"[]{}|\'~

Euroroman

ABCDEFGHIJKLMNOPQRSTUVWXYZ
abcdefghijklmnopqrstuvwxyz
0123456789 □ ± ÿ
!@#%\$%^&*()_+-=,./<>?:':"[]{}|\'~

GDT

ABCDEFGHIJKLMNOPQRSTUVWXYZ

 0123456789 ° ± ø
!@#%\$%^&*()_+-=,./<>?:':"[]{}|\'±°

Monotxt
ABCDEFGHIJKLMNOPQRSTUVWXYZ
abcdefghijklmnopqrstuvwxyz
0123456789 ° ± ø
!@#%&^&*()_+-=,./<>?;':"[]{} \ | ' ~

RomanC (Complex)
ABCDEFGHIJKLMNOPQRSTUVWXYZ
abcdefghijklmnopqrstuvwxyz
0123456789 ° ± ø
!@#%&^&*()_+-=,./<>?;':"[]{} \ | ' ~

RomanD (Duplex)
ABCDEFGHIJKLMNOPQRSTUVWXYZ
abcdefghijklmnopqrstuvwxyz
0123456789 ° ± ø
!@#%&^&*()_+-=,./<>?;':"[]{} \ | ' ~

RomanS (Simplex)
ABCDEFGHIJKLMNOPQRSTUVWXYZ
abcdefghijklmnopqrstuvwxyz
0123456789 ° ± ø
!@#%&^&*()_+-=,./<>?;':"[]{} \ | ' ~

RomandT (Triplex)
ABCDEFGHIJKLMNOPQRSTUVWXYZ
abcdefghijklmnopqrstuvwxyz
0123456789 ° ± ø
!@#%&^&*()_+-=,./<>?;':"[]{} \ | ' ~

Romantic
ABCDEFGHIJKLMNOPQRSTUVWXYZ
abcdefghijklmnopqrstuvwxyz
0123456789 ° ± Ø
!@#%&^&*()_+-=,./<>?;':"[]{} \ | ' ~

ScriptC (Complex)
ABCDEFGHIJKLMNOPQRSTUVWXYZ
abcdefghijklmnopqrstuvwxyz
0123456789 ° ± ø
!@#%&^&()_+-=,./<>?;':"[]{} \ | ' ~*

ScriptS (Simplex)
ABCDEFGHIJKLMNOPQRSTUVWXYZ
abcdefghijklmnopqrstuvwxyz
0123456789 ° ± ø
!@#%&^&()_+-=,./<>?;':"[]{} \ | ' ~*

ABCDEFGHIJKLMNOPQRSTUVWXYZ

!@#\$%^&*()_+-=,./<>?:';:'"[]{}|\|'~

[illegible]

!@#\$%^&*()_+ - = , . / < > ? ; ' : " [] { } \ | ' ~

○ □ △ ◇ ☆ + × * ◀ ▶ ↗ ↘ ⚓ ✂ ♣ ♠ ♡

!@#\$%^&*()_+ - = , . / < > ? ; ' : " [] { } \ | ' ~

$$\aleph' || \pm \mp \times \cdot \div = \neq \equiv \langle \rangle \leq \geq \propto \sim \sqrt{\cup \cap \uparrow \in \rightarrow \uparrow}$$

!@#\$%^&*()_+ - = , . / < > ? ; ' : " [] { } \ | ' ~

36-1-1

!@#\$%^&*()_+ -=,./<>?:';"[]{}|\|'~

— *Journal of the American Medical Association* (1997, 279:1112-1113)

!@#\$%^&*()_+ - = , . / < > ? ; ' : " [] { } \ | ' ~

ABCDEFGHIJKLMNOPQRSTUVWXYZ

!@#\$%^&*() +- = , / < > ? ! ' " [] { } \ | ' ~

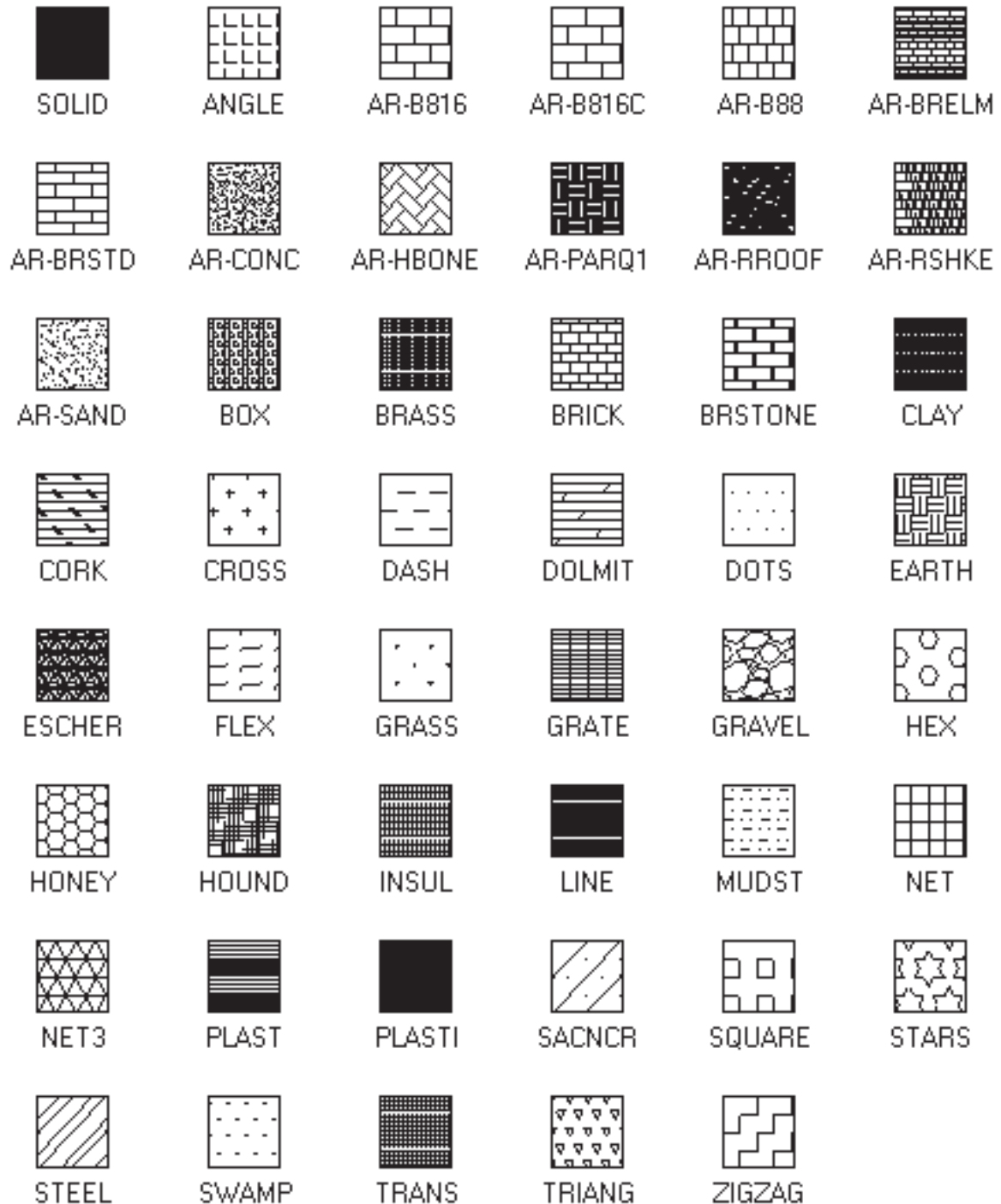
The linetypes shown below are provided with AutoCAD in the *acad.lin* and *acadiso.lin* files. Lintype files are stored in a hidden folder named *C:\Documents and Settings\<user login name>\Application Data\Autodesk\AutoCAD 2010\R18.0\enu\support*, or in a folder defined by the Support File Location item in the Options | Files dialog box.

[illegible]

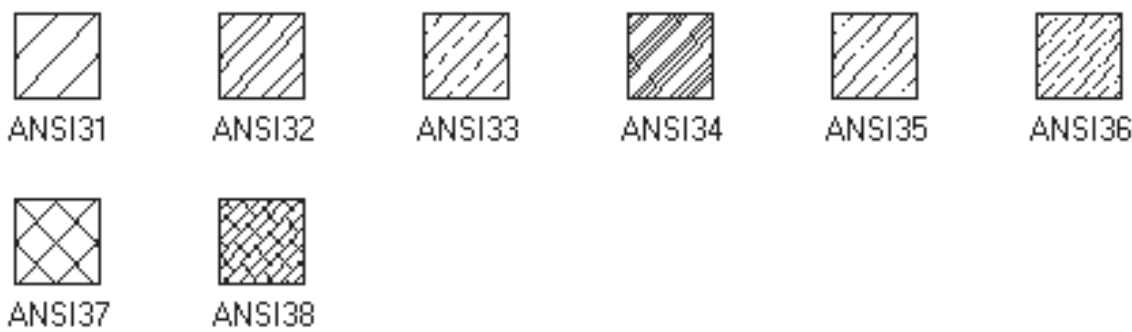
HATCH PATTERNS

The hatch patterns shown below are provided by Autodesk with AutoCAD in the *acad.pat* and *acadiso.pat* files. As well, AutoCAD generates solid-color and gradient fills. Pattern files are stored in a hidden folder named *C:\Documents and Settings\<user login name>\Application Data\Autodesk\AutoCAD 2010\R18.0\enu\support*, or in a folder defined by the Support File Location item in the Options | Files dialog box.

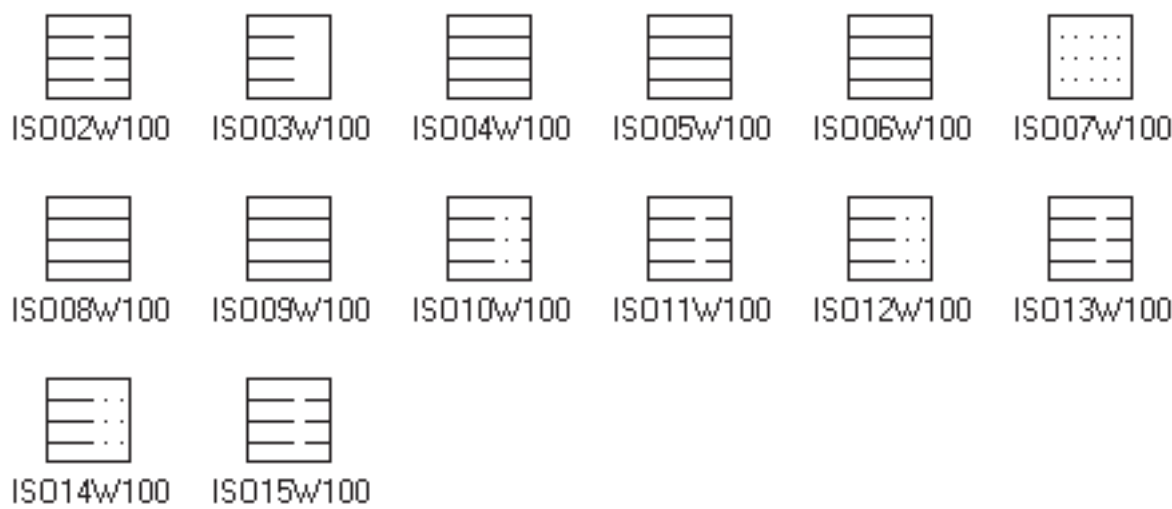
OTHER PATTERNS



ANSI PATTERNS

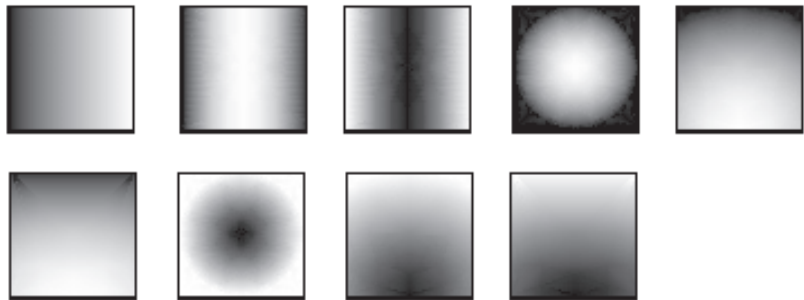


ISO PATTERNS

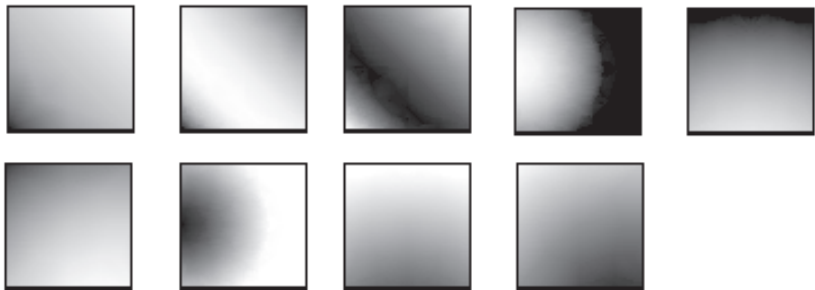


GRADIENT PATTERNS

Centered at 0 Degrees



Off-center at 45 Degrees



LINEWEIGHTS

The lineweights shown below are “hardwired” into AutoCAD; you cannot add, change, or remove them. Lineweights are measured in inches or millimeters.

Millimeters	Sample
0.00 mm*	
0.05 mm	
0.09 mm	
0.13 mm	
0.15 mm	
0.18 mm	
0.20 mm	
0.25 mm	
0.30 mm	
0.35 mm	
0.40 mm	
0.50 mm	
0.53 mm	
0.60 mm	
0.70 mm	
0.80 mm	
0.90 mm	
1.00 mm	
1.06 mm	
1.20 mm	
1.40 mm	
1.58 mm	
2.00 mm	
2.11 mm	

* When lineweight = 0.0, AutoCAD displays the objects at the screen and plotter resolution. For the screen, this is 1 pixel; for the printer, this is 1/dpi resolution (dpi = dot per inch).

DesignCenter, Tools Palette, Symbols & Dynamic Blocks



ADCENTER

The **ADCENTER** command (short for “AutoCAD Design Center”) displays a palette for accessing content in the current and other drawings. DesignCenter provides these services:

- Previews the content of drawing file and raster images.
- Displays layers, linetypes, text styles, table styles, blocks, dimension styles, external references, and layouts (if any) of drawings.
- Accesses blocks from Autodesk’s Web site.
- Copies content from one drawing to another.
- Views the content of folders and files on your computer and network.

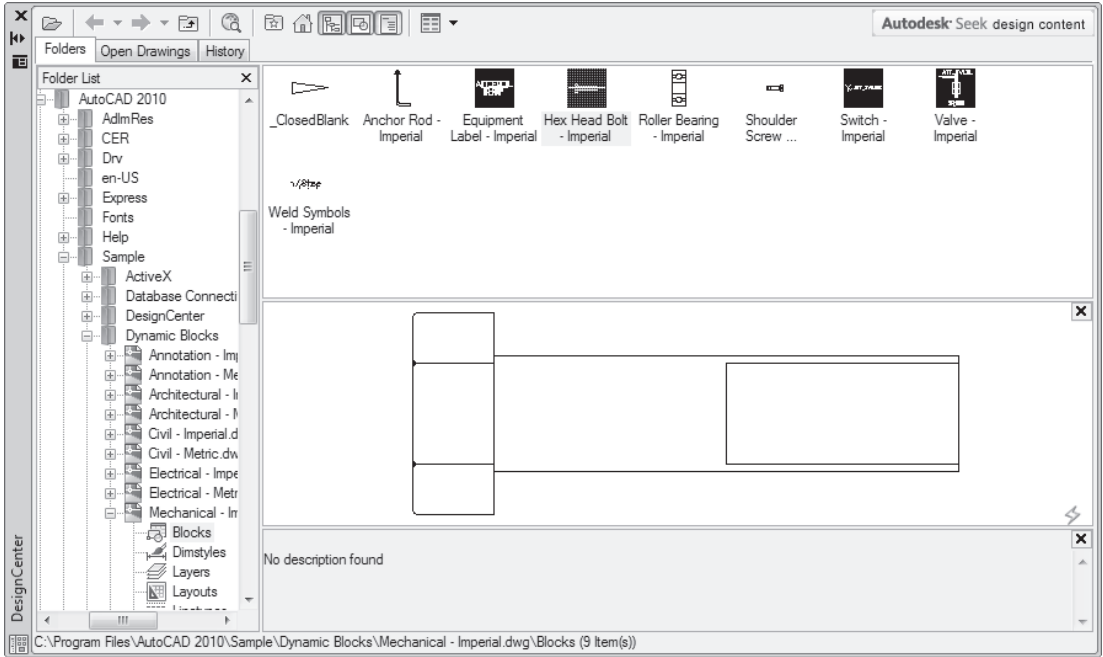
TUTORIAL: DISPLAYING DESIGNCENTER

- I. To display the DesignCenter palette, start the **ADCENTER** command by one of these methods:
 - In the ribbon’s View tab, choose the **DesignCenter** button from the Palettes panel.
 - On the keyboard, press **CTRL+2**.
 - Or, at the ‘Command:’ prompt, enter the **adcenter** command:
Command: **adcenter** (Press ENTER.)
 - Alternatively, enter the aliases **adc**, **dc**, **dcenter**, or **content** at the keyboard.

NEW TO AUTOCAD 2010 IN THIS APPENDIX

- The **SEEK** command accesses blocks and drawings on Autodesk’s Seek Web site.

2. Notice the DesignCenter palette.



The ADCNAVIGATE command specifies the initial path for DesignCenter. This determines which folder is displayed when DesignCenter is first opened:

```
Command: adcnavigate  
Enter pathname <>: (Enter a path, such as c:\drawings.)
```

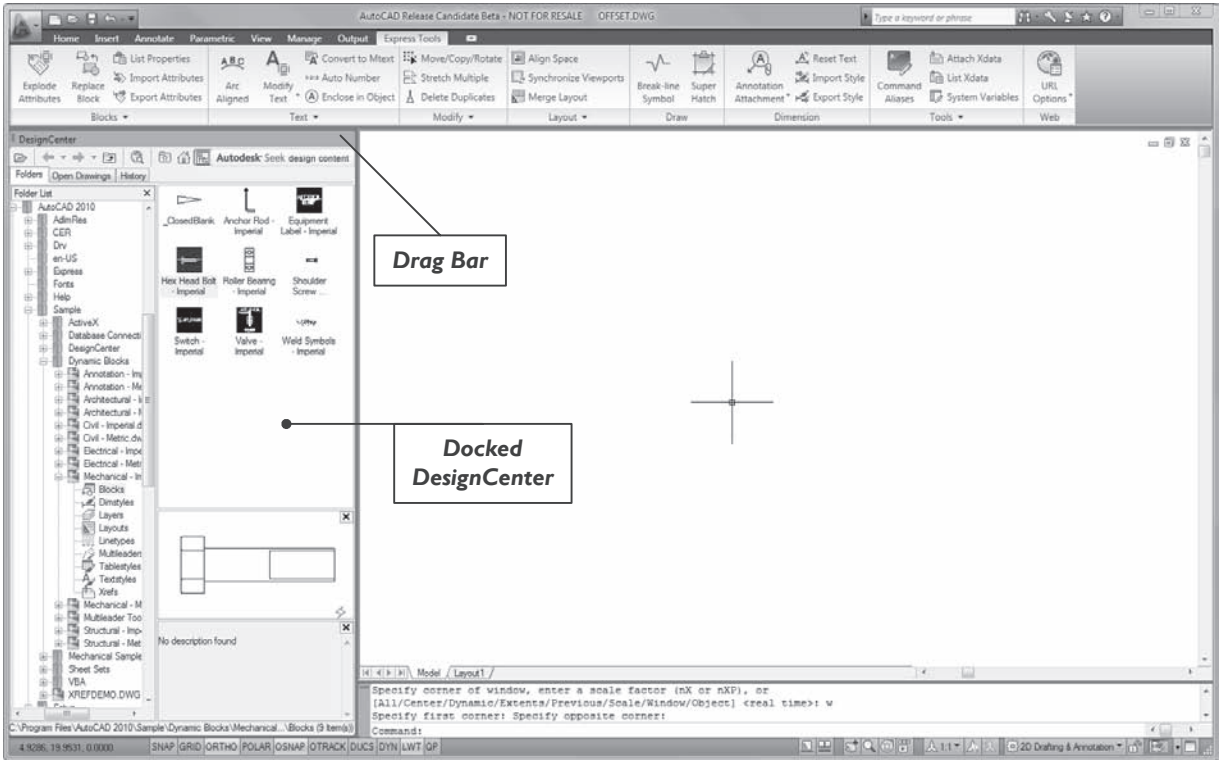
If you want to turn off the DesignCenter palette, use the ADCCLOSE command, or press CTRL+2. As a faster alternative, just click the x in the palette’s upper corner.

NAVIGATING DESIGNCENTER

DesignCenter is a collapsable palette that can float independently of AutoCAD, be docked to the edge of the drawing area, or minimized to just its title bar.

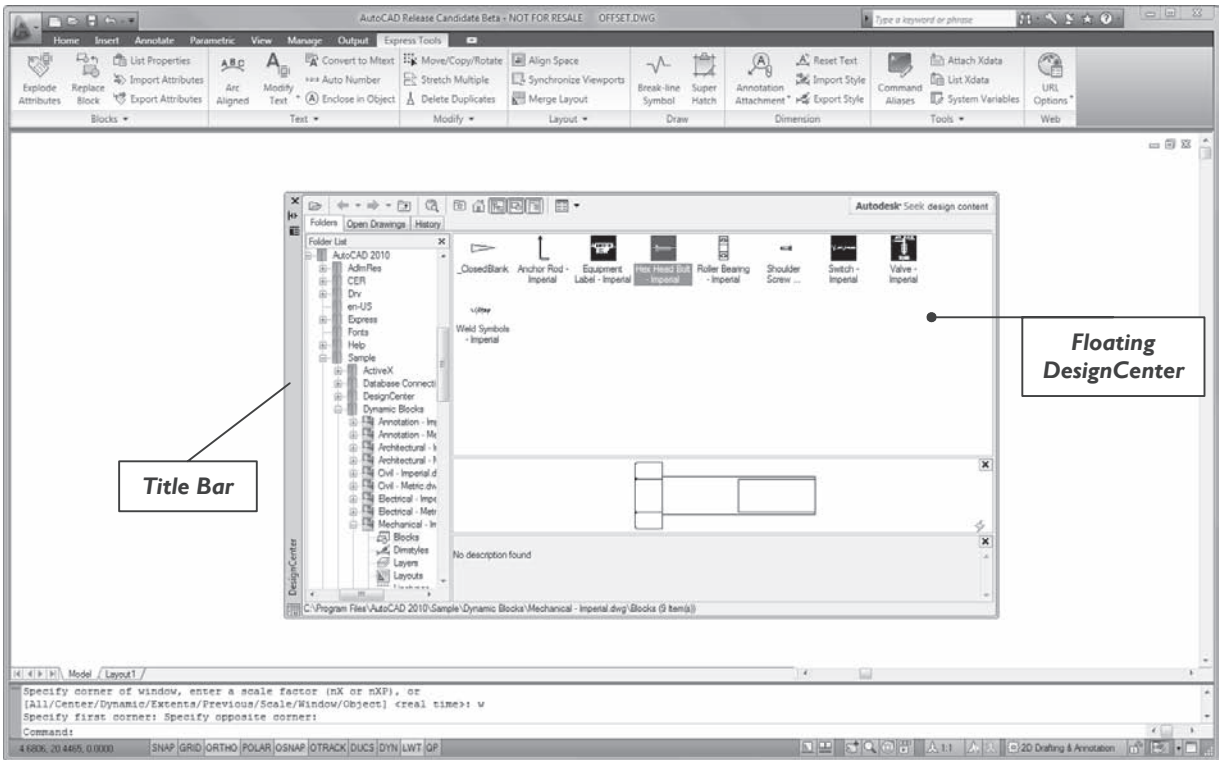
Docking DesignCenter

To dock it, drag DesignCenter by its title bar to an edge of the AutoCAD window. Notice that DesignCenter attaches itself to the edge. The drawback to docking is that the drawing area becomes significantly smaller.



Floating DesignCenter

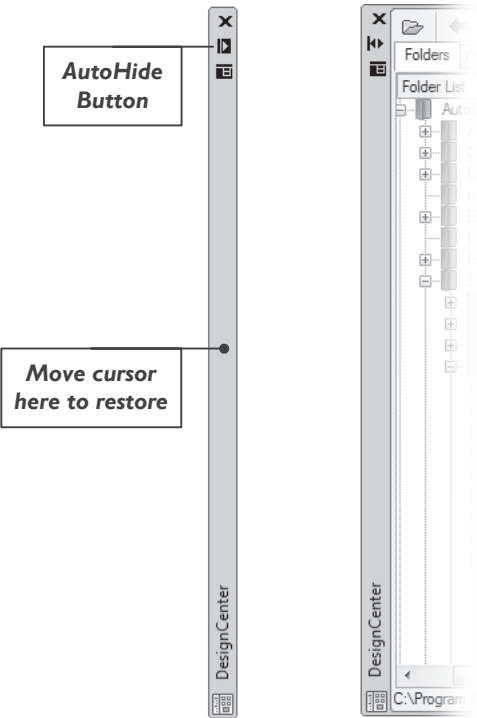
To float, drag the double-line (at the top of DesignCenter) away from the edge of the AutoCAD window. Notice that DesignCenter changes its shape.



You can move DesignCenter anywhere on the screen, including outside of the AutoCAD window. If you have a two monitor system, consider dragging it to the second monitor.

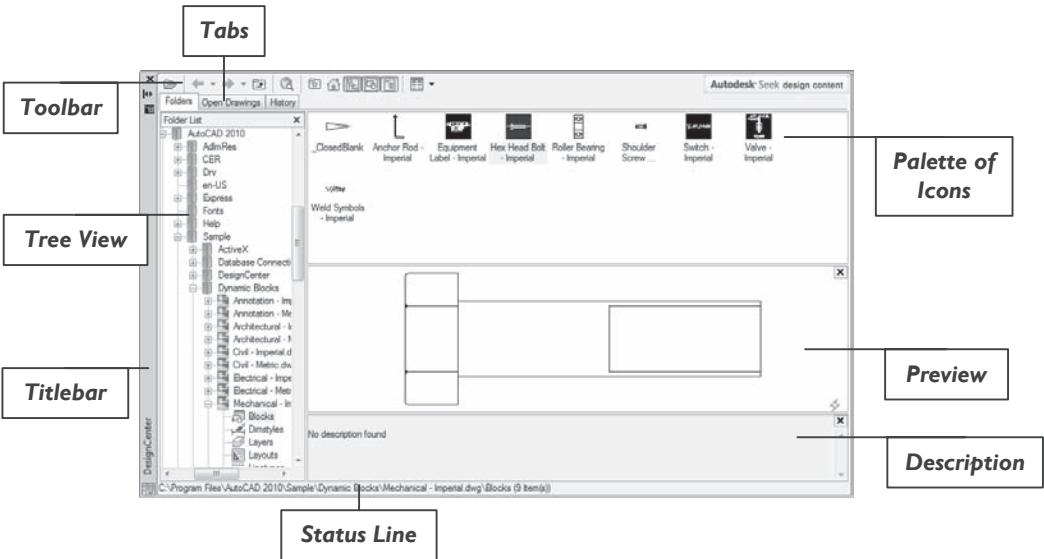
Minimizing DesignCenter

To minimize, click the double-arrow **AutoHide** button on DesignCenter's title bar. Notice that DesignCenter collapses to just its title bar. To restore, pass the cursor over DesignCenter, and it springs back to its original size.



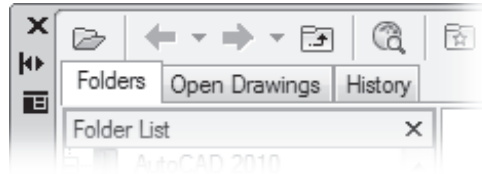
UNDERSTANDING DESIGNCENTER'S USER INTERFACE

The DesignCenter consists of many user interface elements, including toolbar, tabs, title bar, and several panes.



Tabs

DesignCenter employs tabs to display content in several contexts: folder view, open drawings, history of previously-accessed content, and DC design content online.

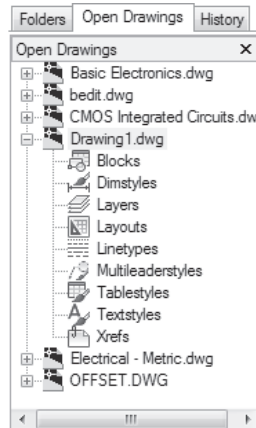


Folders

Click the **Folders** tab to see the names of folders (sub directories) and files on the drives of your computer and network — similar to Windows Explorer.

Open Drawings

The **Open Drawings** tab displays a list of the drawings currently open in AutoCAD. The named content of each drawing is also displayed: blocks, xrefs, layouts, layers, linetypes, table styles, dimension styles, and text styles.

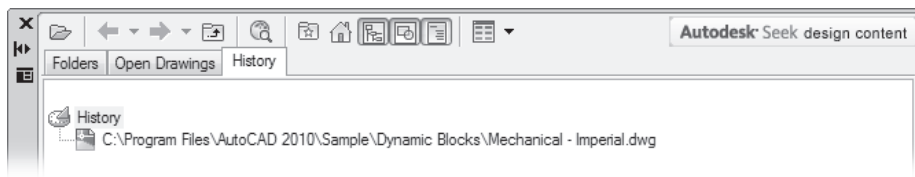


Custom Content

The **Custom Content** tab is present only if a drawing contains proxy data generated by registered (third-party) applications. Either the third-party application must be running or the application must be registered with DesignCenter.

History

The **History** tab displays the last 120 documents that the DesignCenter viewed. Double-click a file name to display its named content (if an AutoCAD drawing) or a thumbnail (if a raster image).



(The DC Online tab was removed from AutoCAD 2010, and was replaced by the Seek button.)

Toolbar

The toolbar changes, depending on the tab you select. For most tabs, the toolbar contains the following buttons:



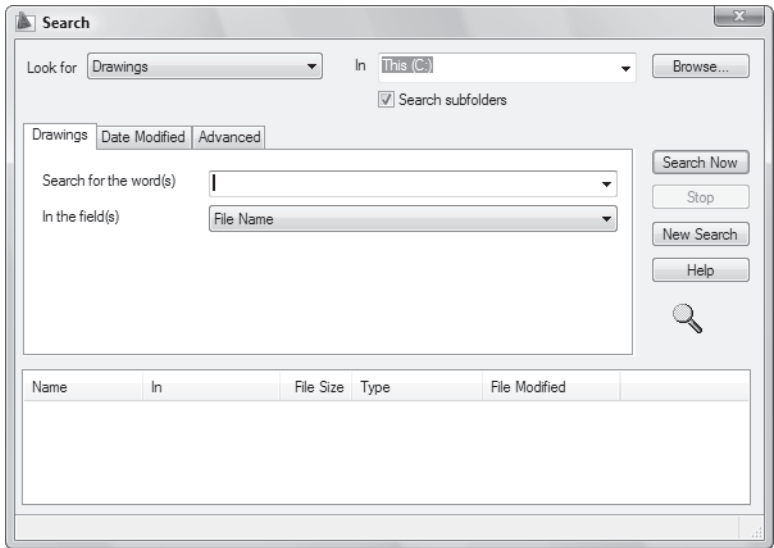
Load displays the Load dialog box for selecting content to load into DesignCenter. You can select content from files located on your computer, from other computers connected with a network, and from Web sites on the Internet.

Back returns to the previous location in the history list. Click the arrow next to this button for the history list.

Forward goes to the next location in the history list.

Up moves up the tree view by one level.

Search displays the Search dialog box, so that you can locate content by specific criteria.



Favorites displays the Windows Favorites folder. Various software applications, including AutoCAD, automatically add files to the Favorites folder. AutoCAD, for example, provides several drawings with blocks that you may find useful for your own drawings. See Appendix I in this book.

Home sends DesignCenter to the designated home folder, which is *C:\Program Files\AutoCAD 2010\Sample\DesignCenter* by default.

Tree View Toggle shows and hides the tree view (the left-hand pane).

Preview shows and hides the preview pane.

Description shows and hides the description pane.

Views changes the display format of the content: large icons, small icons, list view (names only), and detail view (names and file information).



Note: You can force DesignCenter to update its content by *refreshing*. To refresh the display, right-click in the content area, and then select **Refresh** from the shortcut menu.



The Autodesk Seek Design Content button activates the SEEK command. When you click this button, AutoCAD starts your computer's default Web browser, and then opens the Seek web site at seek.autodesk.com.

The site provides blocks and drawings that you can add to your projects. (The related SHAREWITHSEEK command uploads your blocks and drawings to this site for sharing with others.)

In the Autodesk Seek Web page, follow these steps:

- I. Choose a category, such as “Windows.”

2. Select a manufacturer or a product category, such as “aluminum cladded windows.”

Autodesk® Seek

The online source for product specifications and design files

FAQ

Discussions

Feedback

Manufacturer Home

Register

Login

Search

Products by:

Manufacturer

Community

All File Types

Search

Autodesk Seek currently serves the U.S. AEC (architecture, engineering and construction) market

Home >> CSI OmniClass 1.0 >> Openings, Passages, Protection >> Windows >> More

Results Filtered By

Material: Aluminum X

Filter

Product Attributes

All Industry Standard Values

All Material Family Values

All Solar Heat Gain Coefficient

All U Value Values

All Window Style Values

Manufacturer

EFCO Corporation (46)

Kawneer NA (4)

File Type

PDF (49)

DWG (48)

RFA (17)

More

External Catalogs

Manufacturer (46)

Kawneer NA (3)

McGraw-Hill Construction Sweets Networ

Manufacturer

<< < Now showing 1-20 of 50 entries > >>

Email search results

Series 660 Thermal Single Hung, Outside Glazed Sweep Lock Block and Tackle
EFCO Corporation

1 RFA

1 DWG

1 DWF

1 LCF

1 SKP

7 PDF

1 CSV

Details

Series 660 Thermal Single Hung, Sweep Lock Block and Tackle Standard Tall Sill
EFCO Corporation

1 RFA

1 DWG

1 DWF

1 LCF

1 SKP

5 PDF

1 CSV

Details

Series 660 Thermal Single Hung, Outside Glazed Sweep Lock Class 5 Balances
EFCO Corporation

1 RFA

1 DWG

1 DWF

1 LCF

1 SKP

5 PDF

1 CSV

Details

Series 670 Thermal Double Hung, Sweep Locks Class 5 Balances Standard Tall Sill
EFCO Corporation

1 RFA

1 DWG

1 DWF

1 SKP

5 PDF

1 CSV

Details

3. Click on the description to access the download links.

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Search

Products by:

Manufacturer

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All File Types

Search

Autodesk Seek currently serves the U.S. AEC (architecture, engineering and construction) market

Home >> Results >> Series 660 Thermal Single Hung, Outside Glazed Sweep Lock Block and Tackle

Email this page

Series 660 Thermal Single Hung, Outside Glazed Sweep Lock Block and Tackle

12 files for download

Download selected

☒

1 RFA file

Select All

☒

1 DWG file

Select All

☒

1 DWF file

Select All

☒

1 SKP file

Select All

☒

7 PDF files

Select All

☒

1 CSV file

Select All

4. In the right-hand area, choose a file format, such as DWG. Alternatively, you can download the block in all available formats:

Format	Meaning	Useful for AutoCAD
DWG	AutoCAD drawing	Yes
RFA	Revit drawing	No
DWF	Design Web format	No
SKP	SketchUp	No
PDF	Portable Document format	Attach as an underlay
CSV	Comma-separated values	No

5. Click **Download Selected**.
6. Save the file to disk, and then open it in AutoCAD.

(It surprises me that Autodesk's site does not employ AutoCAD's i-drop technology, which allows you to drag blocks from Web sites directly into drawings.)

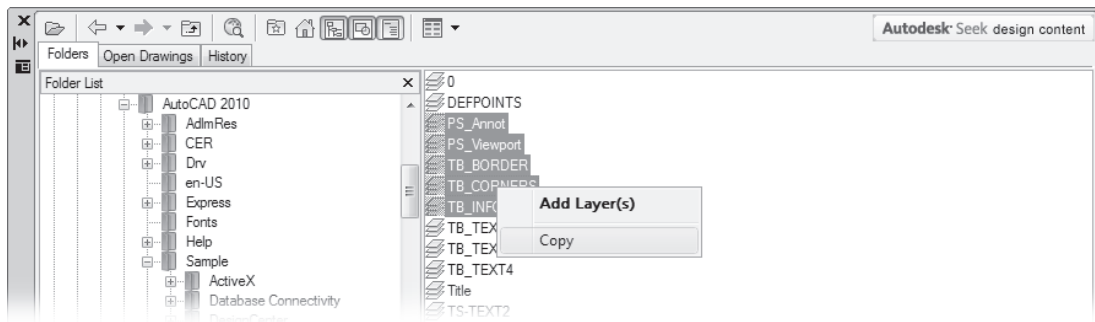
Palette

Right-click the palette (the area displaying the icons) to display shortcut menus. The content of the menus varies according to the items right-clicked.

Copying Content Between Drawings

All palette shortcut menus have one action in common: **Copy** is useful for copying content from one drawing to others. For example, to copy a layer name from one drawing to another, follow these steps:

1. Navigate to the .dwg drawing file with DesignCenter, and then open the *Layers* folder.
2. Right-click the layer you want to copy.
3. From the shortcut menu, select **Copy**.



4. Switch to the drawing to which you want the layer name copied.
5. Press **CTRL+V**. The layer and its properties are added to the drawing.

Dragging Content

There is an alternative method: you can drag content, such as a group of layers, from DesignCenter into drawings:

1. Select the first layer name.
Hold down the **SHIFT** key, and then select another layer name. Notice that DesignCenter highlights all layer names between your two selections.
(To select layers in a nonconsecutive order, hold down the **CTRL** key, and then pick them.)
2. Drag the group of highlighted layer names into the drawing.

AutoCAD copies only the layer names and properties, not objects located on the layers. Sometimes AutoCAD warns:

Duplicate names ignored.

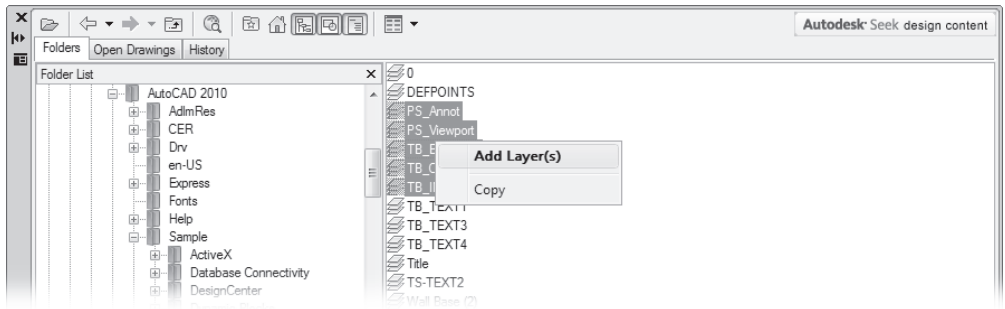
This means that if a layer of that name already exists, AutoCAD does not duplicate it.

Adding Content

The second item on the shortcut menu varies, depending on the content; many have **Add**, such as Ass Layer(s). This option adds the content to the current drawing.

It may seem to you that the **Copy** and **Add** options are similar in purpose. The difference is that **Copy** copies the content to the Clipboard. From there you have to use the **PASTE** command to get the data into the drawing. (This allows you to move content to another CAD system, such as AutoCAD LT.)

In contrast, **Add** immediately adds the content to the current drawing.

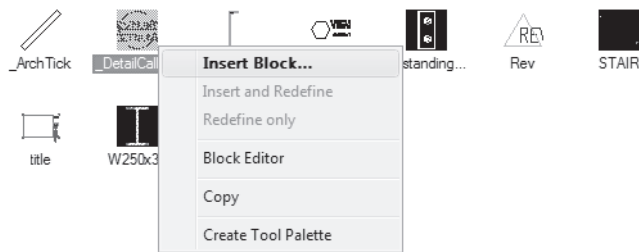


For example, to add layers to the current drawing, right-click the names, and then select **Add Layer(s)** from the shortcut menu. As before, AutoCAD warns that duplicate names are ignored.

Working with Blocks and Xrefs

The shortcut menus for blocks and external references operate differently from other types of content.

Blocks. When you right-click blocks, the shortcut menu includes the **Insert Block** option, which displays the Insert dialog box. It is identical to the Insert dialog box described later in this chapter.



Insert and Redefine. When you try to add a layer or linetype that already exists in the drawing, AutoCAD ignores the command, and doesn't add it; blocks are handled differently, however. When you add a block from DesignCenter to a drawing that already contains a block of the same name, the **Insert and Redefine** and **Redefine Only** options update blocks in the drawing with the definition you are adding.

Block Editor. Selecting the **Block Editor** option opens the block in the Block Editor. This allows you to edit the block.

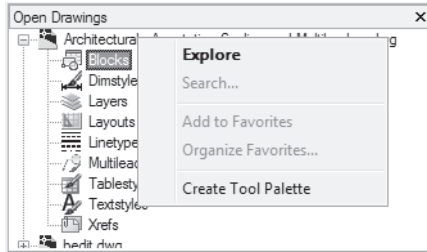
External References. Selecting the **Attach Xref** option display

Create Tool Palette

The **Create Tool Palette** option adds the selected object to the Tool palette.

Tree View

Right-click the tree view to display a shortcut menu that mimics the functions of some buttons on the toolbar: Find and Favorites.



Explore has the same effect as clicking the small + symbol next to a folder or file name.

Search brings up the Search dialog box to hunt down content.

Add to Favorites adds the drawing or folder to the list of “favorites,” which acts like a bookmark.

Organize Favorites displays the Windows folder showing the *C:\Users\<login name>\Favorites\Autodesk* folder.

Create Tool Palette adds *all* of the drawing’s blocks to the Tool palette.

Set as Home is the same as using the ADCNAVIGATE command.

Description

Right-click the description area to display a cursor menu with commands suitable for copying the text to the Clipboard.

Title Bar

Double-click the title bar to dock DesignCenter on the right side of the AutoCAD window.

TOOLPALETTES

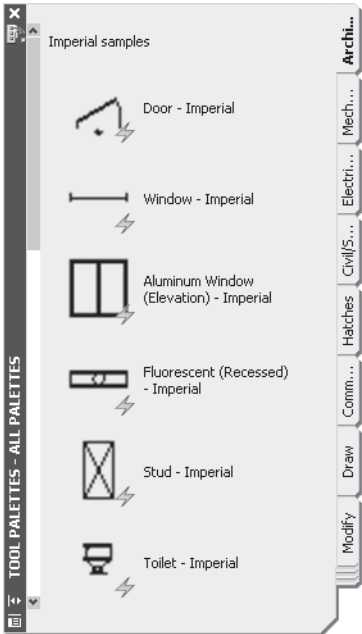
The **TOOLPALETTES** command displays palettes that provide access to frequently-used content.

The Tool palette window is a centralized collection of blocks, hatches, objects, materials, and commands that you use often. It eliminates the need to hunt through DesignCenter, the Hatch and Gradient dialog box, the Materials palette, and the Insert Block dialog box for specific blocks, hatch patterns, and materials. The Tool palette can also include programming routines and scripts.

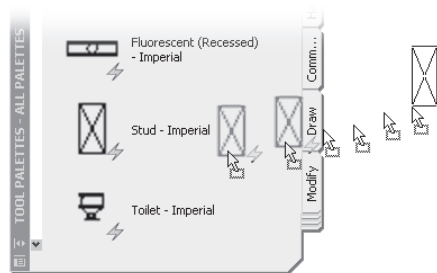
TUTORIAL: DISPLAYING AND USING THE TOOLPALETTE

1. To display the Tool palette, use the **TOOLPALETTES** command with one of these methods:
 - In the ribbon's View tab, choose the **Tool Palettes** button from the Palettes panel.
 - On the keyboard, press **CTRL+3**.
 - Or, at the 'Command:' prompt, enter the **toolpalettes** command:
Command: **toolpalettes** (Press ENTER.)
 - Alternatively, enter the **tp** alias at the keyboard.

Notice the Tool palette has numerous tabs. (The number and names of tabs depend on how the palette was last used, and which workspace AutoCAD is in.)



2. The *tabs* along the side access the palettes. Click a tab to view the associated palette. Each palette contains one or more *tools*. A tool can be a hatch pattern, a block, a material, an object, or a command.
3. To place a block (or any other object) in the drawing, drag it from the palette into the drawing. AutoCAD places the block at the point where you let go of the mouse button; use object snaps to place the block accurately. There are no prompts to answer; the block is scaled automatically to its preset units.



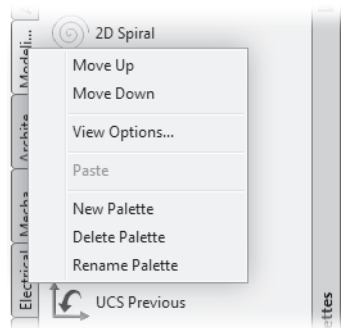
To turn off the Tool palette, press **CTRL+3** or enter the **TOOLPALETTECLOSE** command. As an alternative, click the x in the palette’s upper corner.

NAVIGATING THE TOOL PALETTES: ADDITIONAL METHODS

AutoCAD hides many of the Tool palette’s options in shortcut menus, which differ depending on which part of the palette you right-click.

Right-click Commands: Tabs

Right-click the tab of the current palette (the one “on top”) to see the following commands:



Command	Comment
Move Up	Moves the palette up by one position.
Move Down	Moves the palette down by one position.
View Options	Displays the View Options dialog box for changing the size and style of icons and text.
Paste	Pastes data from the Clipboard into the palette. This option is available only when the Clipboard contains appropriate data, such as an AutoCAD block. You can copy a block in DesignCenter for pasting in the tool palette, but not from AutoCAD itself, because in AutoCAD you never see the block <i>definition</i> ; you only see insertions of it.
New Tool Palette	Creates a new blank palette, and prompts you to name the tab.
Delete Tool Palette	Warns against deleting the palette, and then deletes it when you answer OK .
Rename Tool Palette	Renames the tab.

View Options Dialog Box

The **View Options** option displays the View Options dialog box for changing the size and style of icons and text.

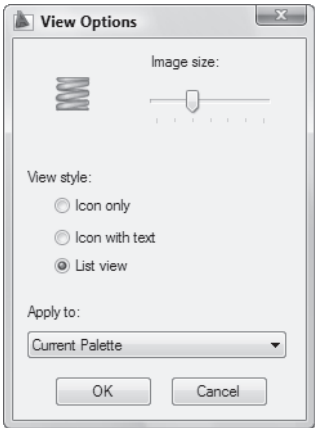
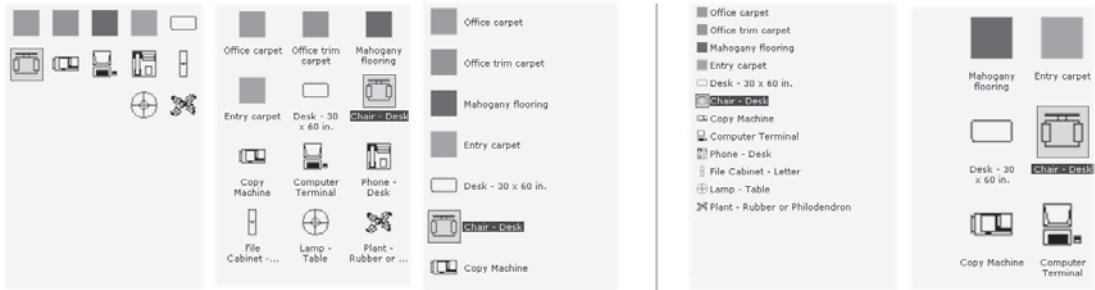


Image Size changes the size of icons from small (14 pixels) to large (54 pixels).

View Style changes the display of icons and text:

- **Icon only** displays icons only.
- **Icon with text** displays icons and text.
- **List view** displays small icons with text.

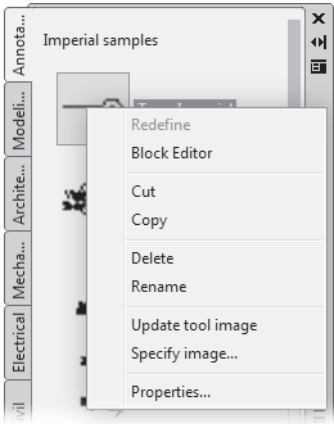


Left to right: Icons only; icons with text, list view, small images, and large images.

Apply to determines whether the changes apply to the current palette (tab), or to all palettes (tabs).

Right-click Commands: Icons

Right-click any icon (called a “tool” by Autodesk) to get the following shortcut menu:

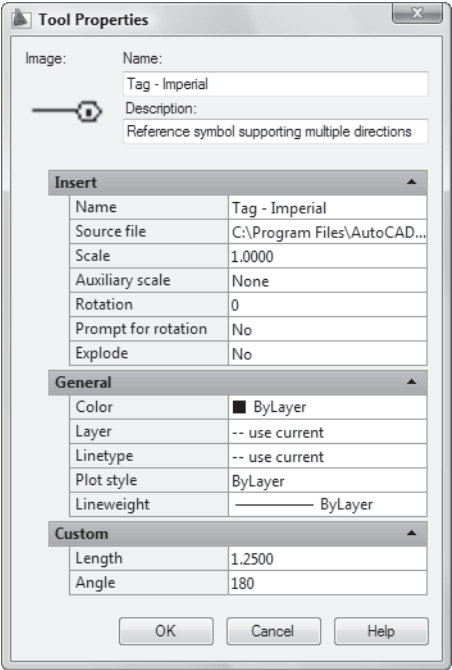


Command	Comment
Redefine	(Blocks only) Redefines the block, if the definition has changed.
Block Editor	(Blocks only) Opens the block in the Block Editor.
Cut	Removes the tool and places it in the Clipboard.
Copy	Copies the tool to the Clipboard.
Delete	Removes the tool.
Rename	Renames the tool.
Update Tool Image	(Blocks only) Forces AutoCAD to regenerate images, should the block have changed.
Specify Image	Selects a raster image for the tool's icon.
Properties	Displays the Properties dialog box for changing the properties of the hatch and block tools. This palette has content similar to the one displayed by the PROPERTIES command.

Tool Properties Dialog Box

The Tool Properties dialog box displays everything AutoCAD knows about the object. Shown below is the list of properties for the selected block. For example, you can change the name of the block, its description, scale, and whether it is inserted exploded (**Explode**).

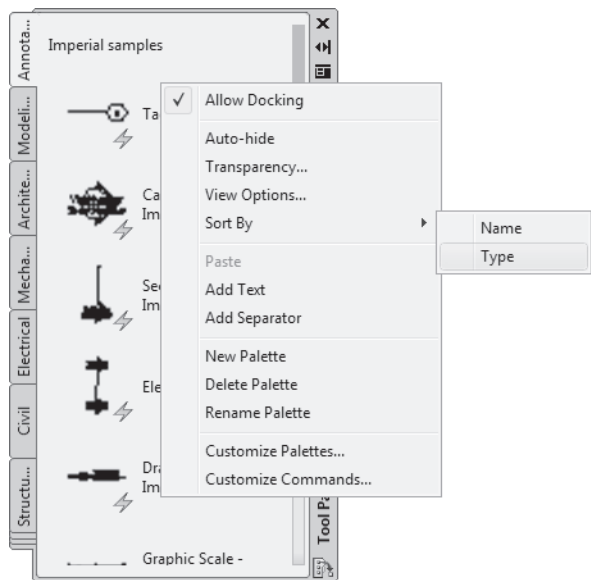
In particular, you can define a target layer through the Layer property: change “-- use current --” to another name, and the block is inserted on that layer, instead of the current one.



Properties shown in white can be changed; gray ones cannot. Those properties listed under Custom are specific to dynamic blocks.

Right-click Commands: Palettes

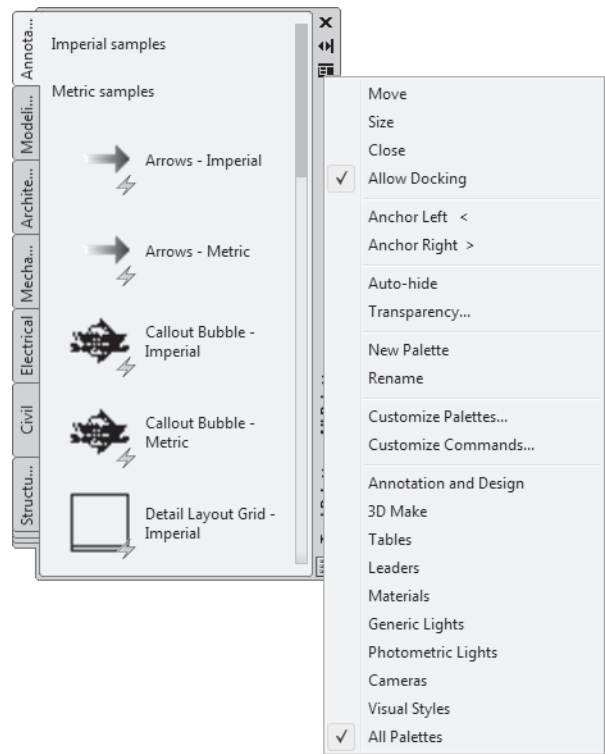
Right-click anywhere on an unused area of the palette — not on an icon or a tab — to get the following shortcut menu:



Command	Comment
Allow Docking	Determines whether the tool palette docks against the side of the AutoCAD window when it gets close enough.
Auto-Hide	Toggles whether the tool palette reduces to just the title bar when the cursor leaves the palette.
Transparency	Displays the Transparency dialog box to make the tool palette “see thru.”
View Options	Displays the same dialog box as on the tab’s shortcut menu.
Sort By	Sorts icons by name or type of object.
Paste	Pastes the tool; command is available only when a tool has been copied or cut to the Clipboard.
Add Text	Adds a line of text to describe a group of icons.
Add Separator	Adds a horizontal line to separate groups of icons visually.
New Palette	Adds a new blank palette.
Delete Palette	Removes the current palette.
Rename Palette	Changes the name of the current palette.
Customize Palettes	Displays the Customize dialog box with the Tool Palettes tab.
Customize Commands	Displays the Customize User Interface dialog box (CUI).

Title Bar

When you right-click the title bar, you can access all the palette groups and “covered” tabs:

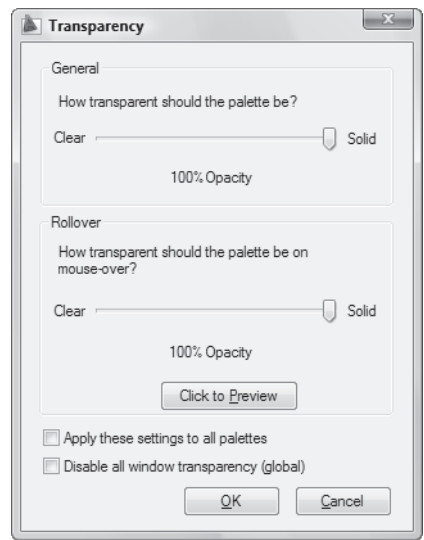


In the lower portion of the shortcut menu are the names of palette groups, such as Tables and Leaders. If you cannot find a group name among the tabs, this is the place to look.

The following sections describe the more important options found on this shortcut menu.

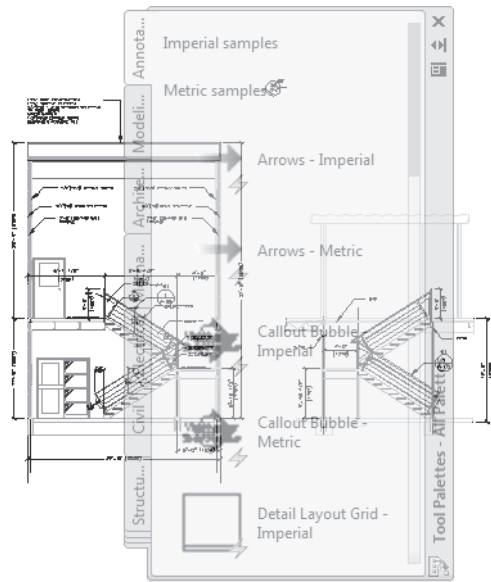
Transparency

The Transparency option changes the transparency of tool palettes:



Moving the pointer toward Clear makes the tool palettes more opaque, while moving it toward Solid makes them more translucent. Transparency is unavailable when AutoCAD graphics are hardware-accelerated.

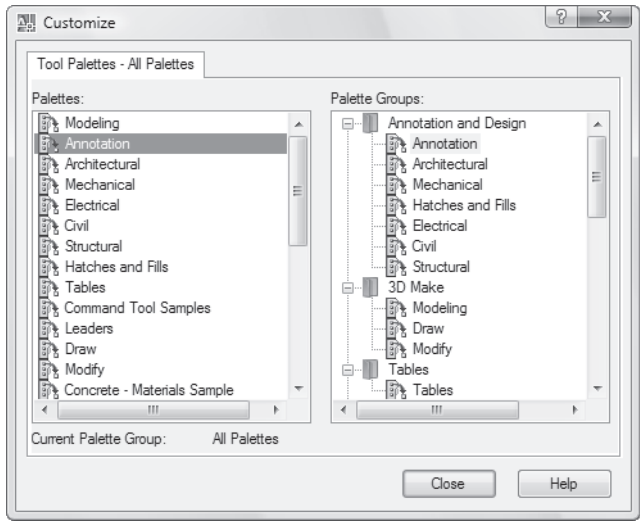
Transparency lets you see both the tool palettes and the drawing.



Whether or not you use transparency is a matter of personal preference. I find transparency makes it harder to see both the palette and the drawing, so I keep it turned off; others like being able to see both at once.

Customize Palettes

The Customize Palettes option runs the CUSTOMIZE command for organizing palettes by order and by group.



The left pane shows all palettes; the right pane allows you to segregate palettes into groups. When you make a group current, only the palettes in the group are displayed by the Tool palette. Right-click a group, and then select **Make Current** from the shortcut menu.

Drag palettes up and down to change their order. To create a new (blank) palette, right-click, and then select **New Palette** from the shortcut menu. To delete or rename a palette, right-click it and then select the appropriate command from the shortcut menu. Deleting palettes is permanent; there is no undo.

You can export selected palettes to share with other AutoCAD users. Right-click a palette name, and then select **Export**. Notice that AutoCAD prompts you to save the palette as a *.xtp* file. Use the **Import** button to read tool palettes created by others. The source drawing files must accompany blocks sent with this export and import process.

Customize Commands

The **Customize Commands** option runs the CUI command, which lets you add commands by dragging them from the Customize User Interface dialog box onto the Tool palettes.

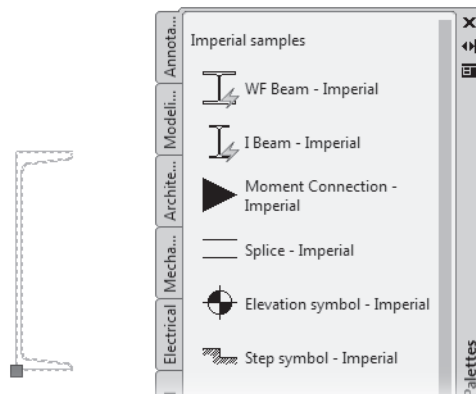
COMMAND TOOLS BY EXAMPLE

Command tools are made from geometric objects, such as lines, polylines, and dimensions — any object other than blocks, xrefs, materials, images, hatches, or gradient fills. You create tools by dragging objects onto the Tool palette. Autodesk calls this feature “tools by example,” because the tools have the same properties as the original objects, including layer, color, and lineweight.

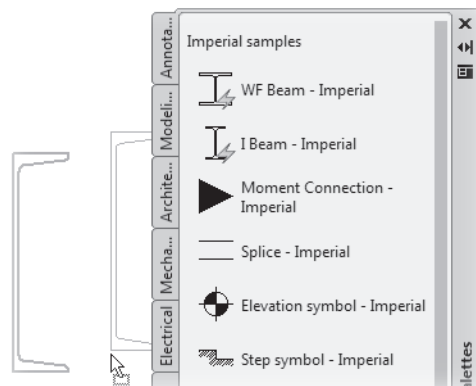
(When you drag the command tool from the Tool palette into the drawing, AutoCAD executes the command associated with the tool.)

To create command tools, drag objects onto the palette. Follow these steps:

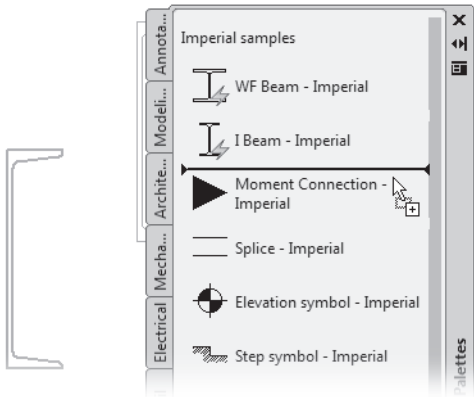
1. Select the object. As usual, handles appear.



2. Select the grip, and then hold down the **right** mouse button until the handle disappears.



3. Drag the object onto the palette. The thick horizontal line helps you position the tool.

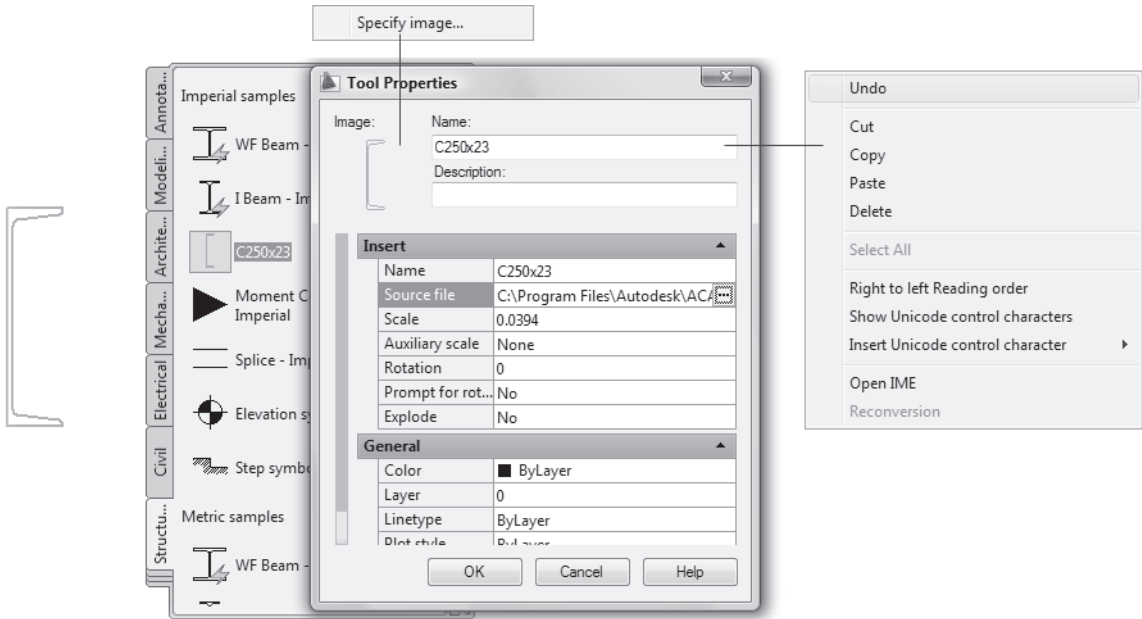


(Once the object is on the palette, AutoCAD sometimes adds a flyout that shows all other drawing commands — but these flyouts are only for commands that create objects and dimensions, and not blocks. The flyout lets you place other objects on the same layer.)

In addition to storing objects as tools, palettes remember the object’s properties, such as layer, color, and linetype. The tools have the same properties as the objects selected from drawings, including the layers from the source object, if the layers do not exist in the drawing.

To change their properties, select one or more tool icons. (To select more than one, hold down the CTRL key while selecting them.) Right-click them, and then select **Properties** from the shortcut menu.

The Tool Properties dialog box has a number of commands that are accessed through shortcut menus:



Changing the Image

AutoCAD automatically includes an image appropriate to the object. To change the image, right click the icon, and then select **Specify Image**. In the Specify Image File dialog box, select a bitmap (.bmp), JPEG (.jpg), PNG (.png), GIF (.gif), or TIFF (.tif) file, and then click **Open**. I have found that the size does not matter: AutoCAD resizes the image to fit.

You cannot change the images for hatches, tables, or blocks.

Changing the Flyout (Objects and Commands)

In most cases, AutoCAD automatically adds the flyout. If it doesn't, change the **Use Flyout** option from **No** to **Yes**.

The flyout consists of the objects illustrated by the figure above. Click **Flyout Options** to reduce the objects displayed on the flyout. You cannot add flyouts to hatches, tables, materials, or blocks.

Auxiliary Scale (Blocks and Hatches)

The **Auxiliary Scale** option applies to blocks and hatches. This is a scale factor that multiplies the dimension scale (as set by the DIMSCALE system variable) and the plot scale (as set by the PAGESETUP command) as follows:

Auxiliary Scale	Scale Factor is...
None	Unchanged.
Dimscale	Multiplied by the value stored in DIMSCALE sysvar.
Plot Scale	Multiplied by the value set by the PAGESETUP command's Plot Scale factor.

Prompt for Rotation (Blocks)

When on, the **Prompt for Rotation** option prompts you for the rotation angle when you drag blocks into the drawing. When off, it does not; blocks are placed at 0 degrees rotation.

ADDITIONAL CONTENT

Tool palettes also handle additional content, as described here.

Drawings and Images

In addition to dragging objects from drawings onto palettes, you can also drag drawing files (.dwg) and raster image files (such as .jpg and .tif) onto palettes from Windows Explorer. Drawing files are placed as blocks, while raster files are placed as images.

Command Tools

Commands, macros, AutoLISP and Visual LISP expressions, VBA macros, and scripts can be placed on palettes. The process, however, is not as simple as dragging and dropping objects. For programs and macros, you need to follow this indirect method:

1. Place a geometric object (tool) on a palette.
2. Right-click the tool, and then select **Properties** from the shortcut menu.
3. If necessary, turn off **Use Flyout** (set to **No**).
4. In the Command String area, replace the command with the programming code or macro.
You can either type the new code, or copy and paste it from another source.

To change the icon for programs like this: (1) right-click the image, (2) from the shortcut menu, select **Specify Image**, and (3) select another image.

Sharing Tool Palettes

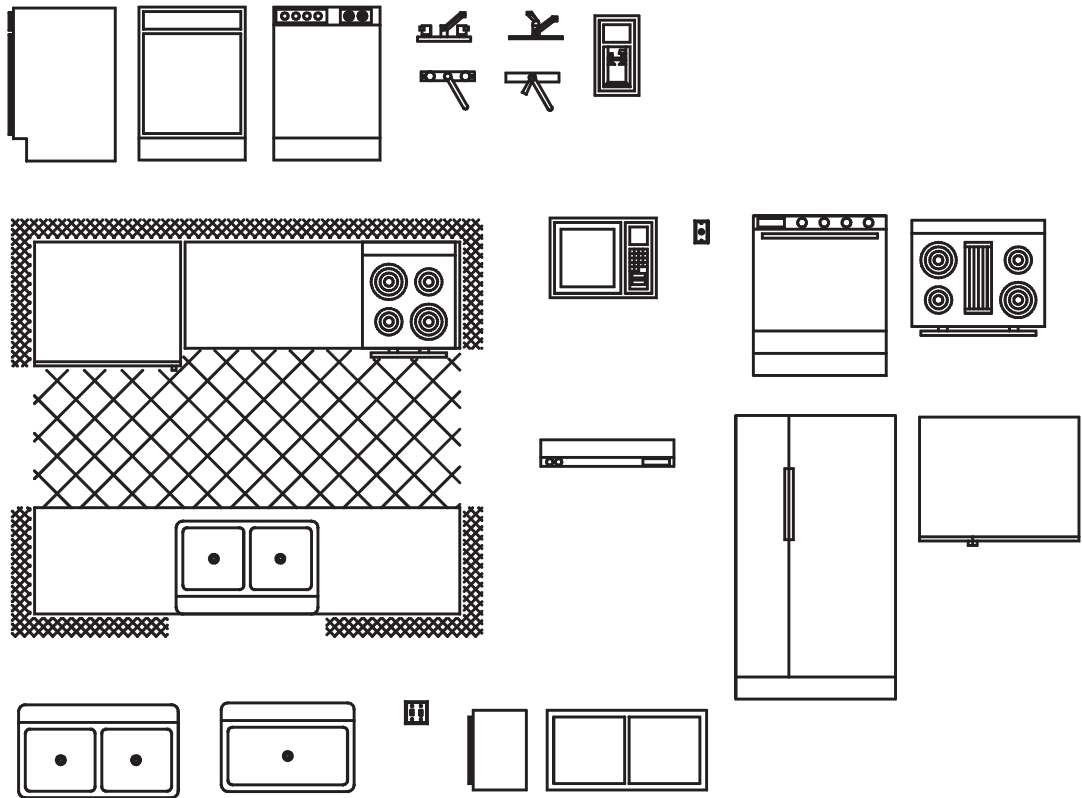
In addition to dragging objects from DesignCenter into drawings, you can also drag them onto the Tool Palettes. Here is a summary of the dragging operations you can perform:

- From DesignCenter onto the Tool Palettes (but not from the Tool Palettes onto DesignCenter).
- From the Tool Palettes onto the drawing, and in AutoCAD from the drawing onto the Tool Palettes.
- From DesignCenter onto the drawing (but not from the drawing onto DesignCenter).

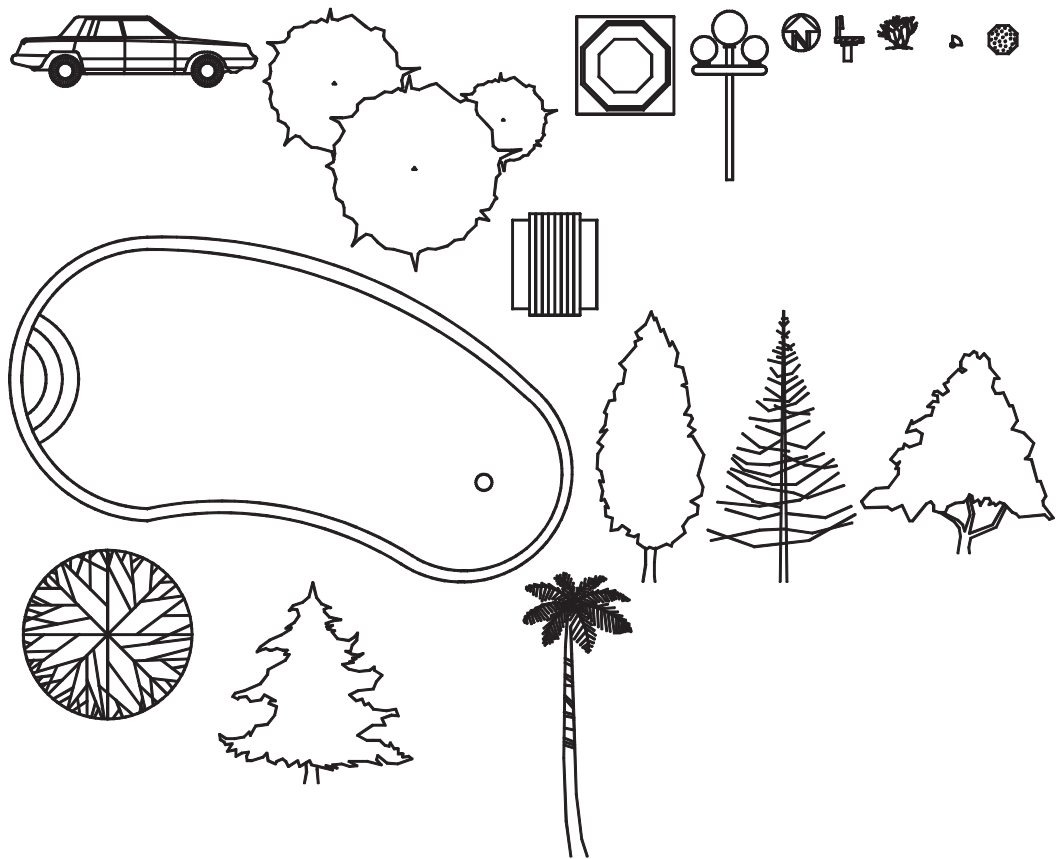
DESIGNCENTER BLOCKS

The symbols (blocks) shown on the following pages are provided by Autodesk with AutoCAD, and can be accessed with DesignCenter. These DWG-format files are stored in the `\autocad 2010\sample\designcenter` folder, and are often available in metric or Imperial versions.

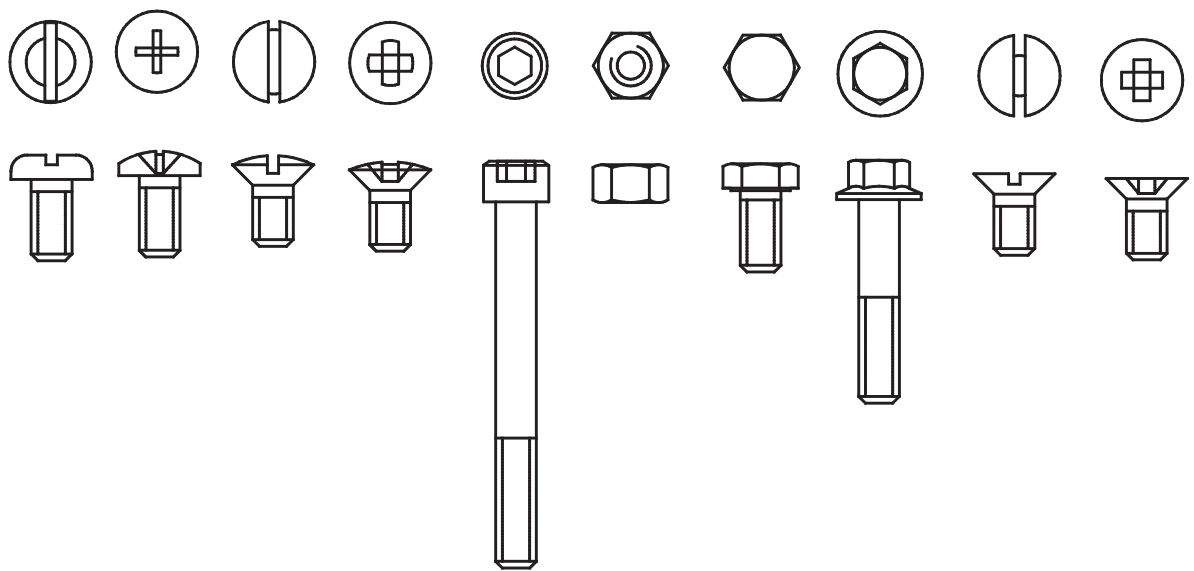
AEC: KITCHENS



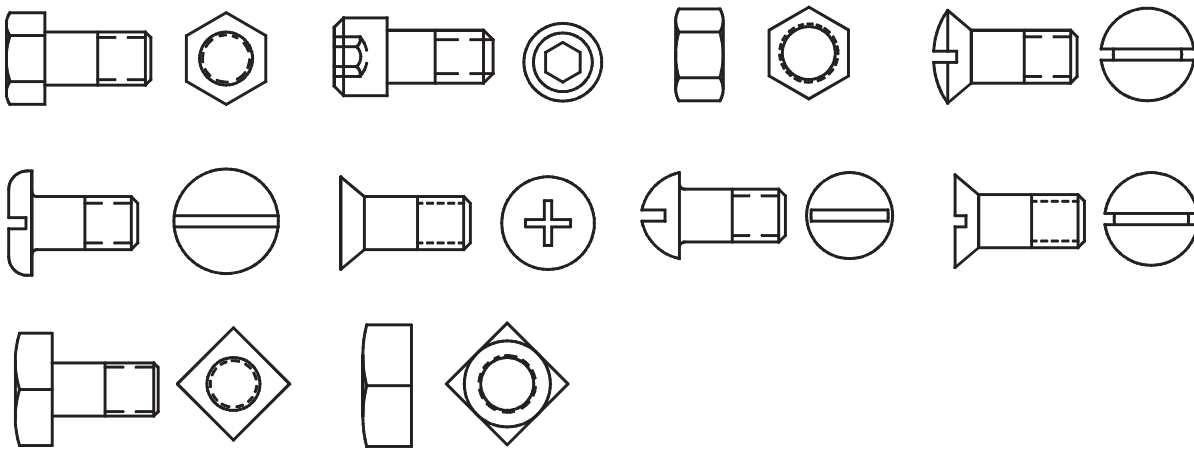
AEC: LANDSCAPING



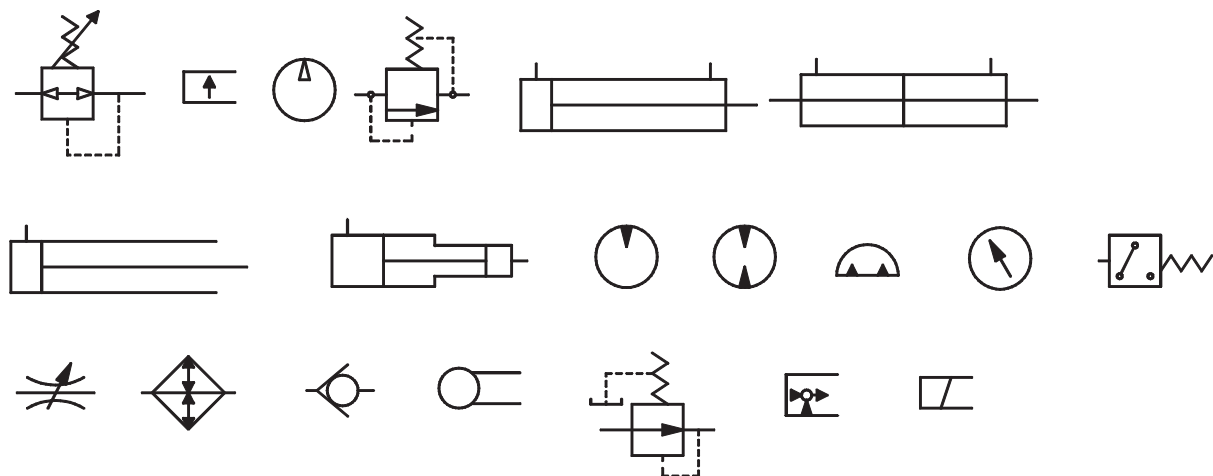
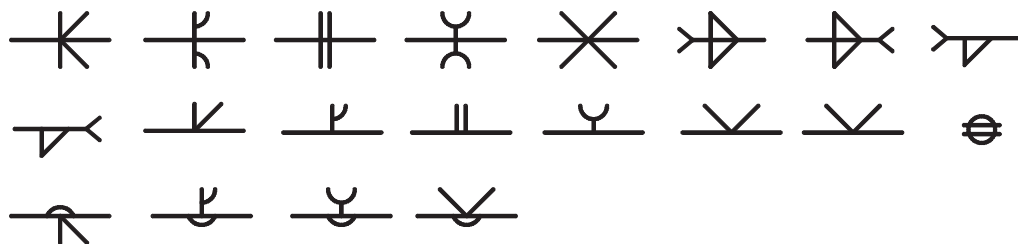
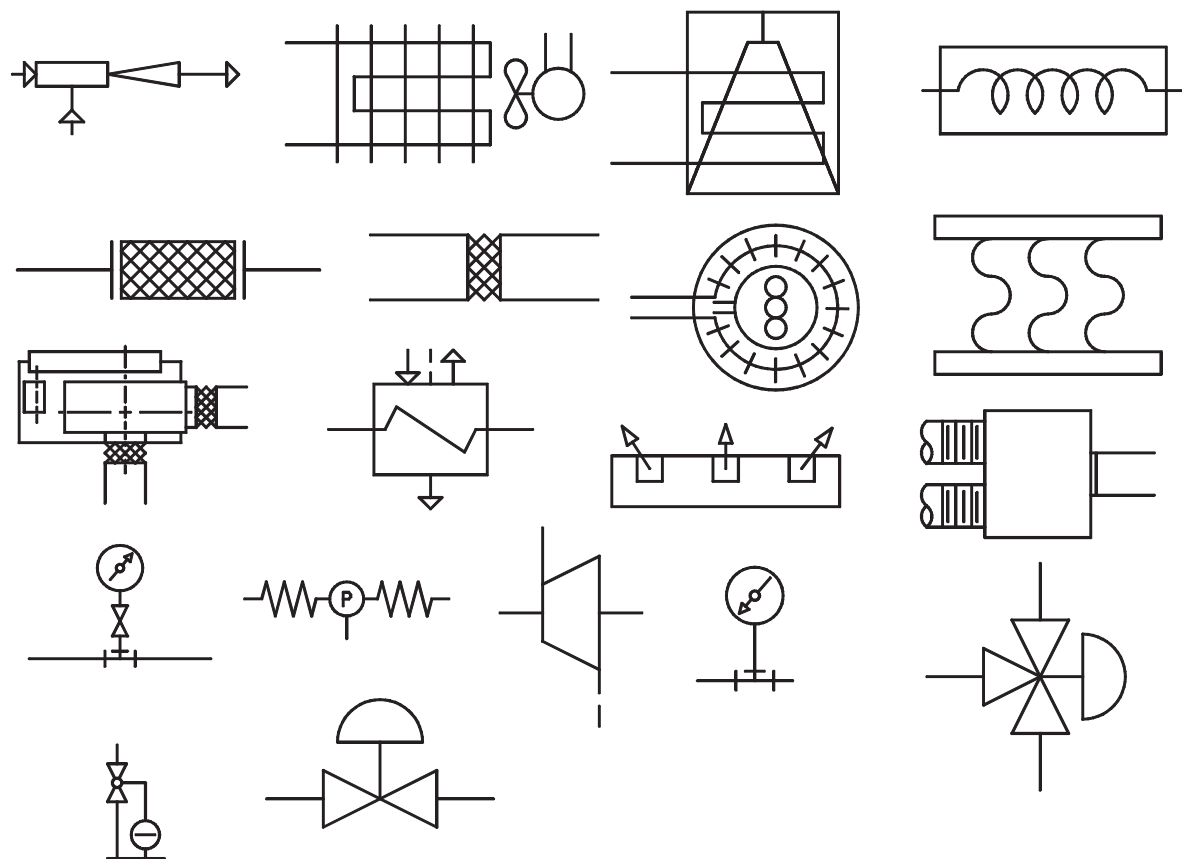
MECHANICAL FASTENERS - METRIC



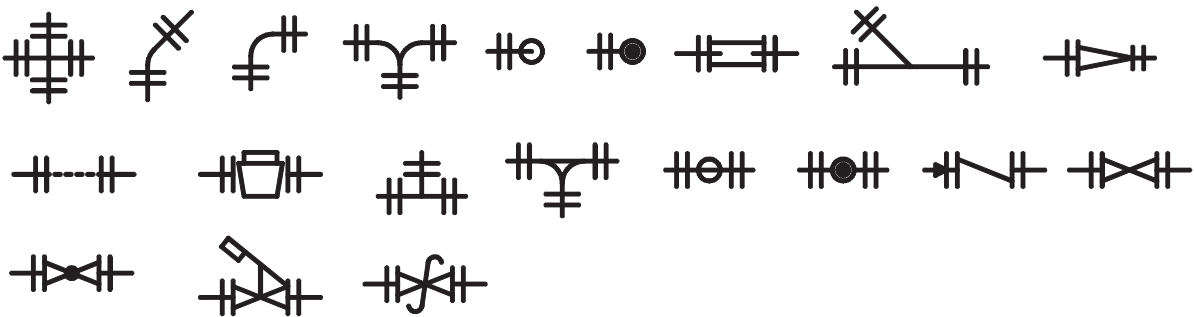
MECHANICAL FASTENERS - US



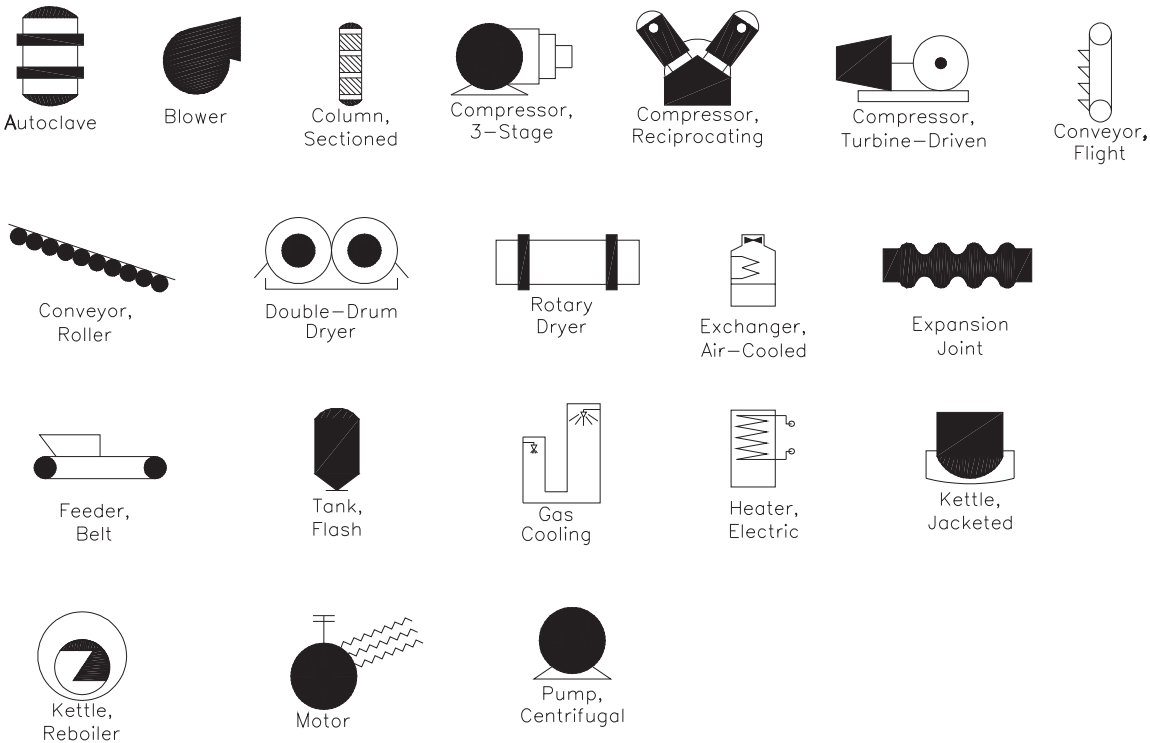
MECHANICAL HYDRAULIC - PNEUMATIC



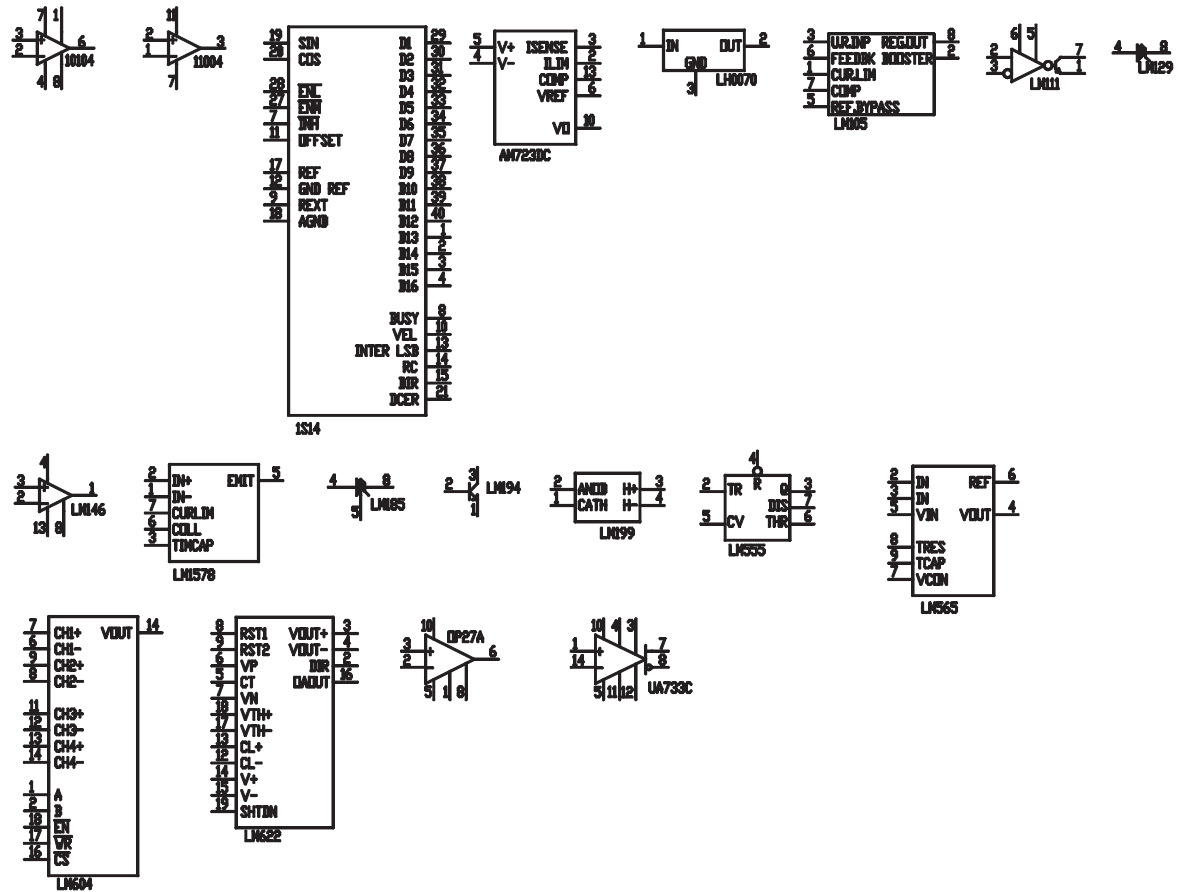
MECHANICAL PIPE FITTINGS



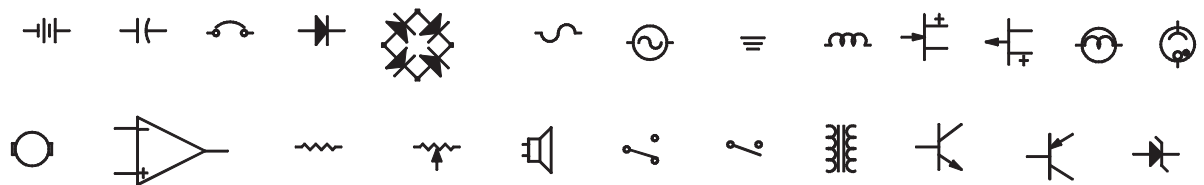
MECHANICAL PLANT PROCESS



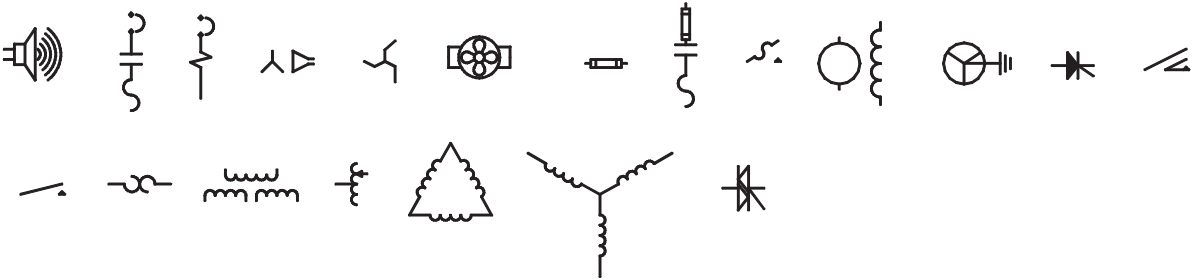
ELECTRONICS: ANALOG INTEGRATED CIRCUITS



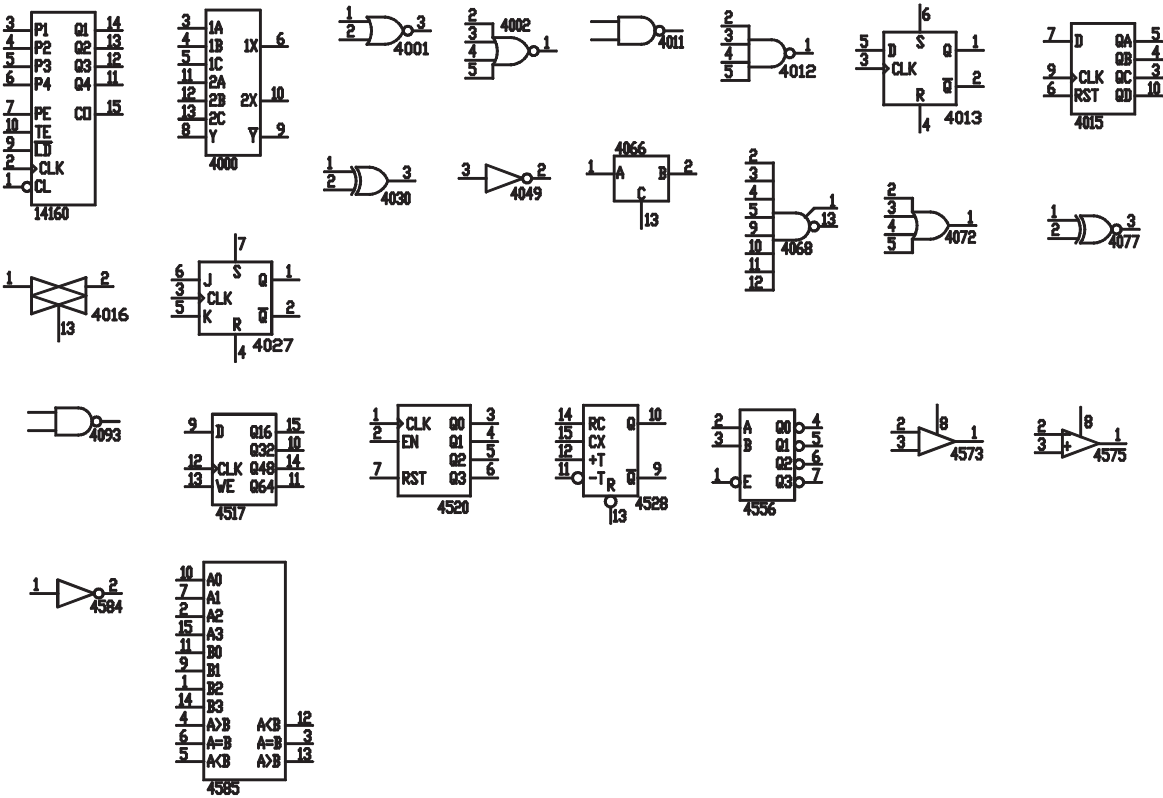
ELECTRONICS: BASIC ELECTRONICS



ELECTRONICS: ELECTRICAL POWER



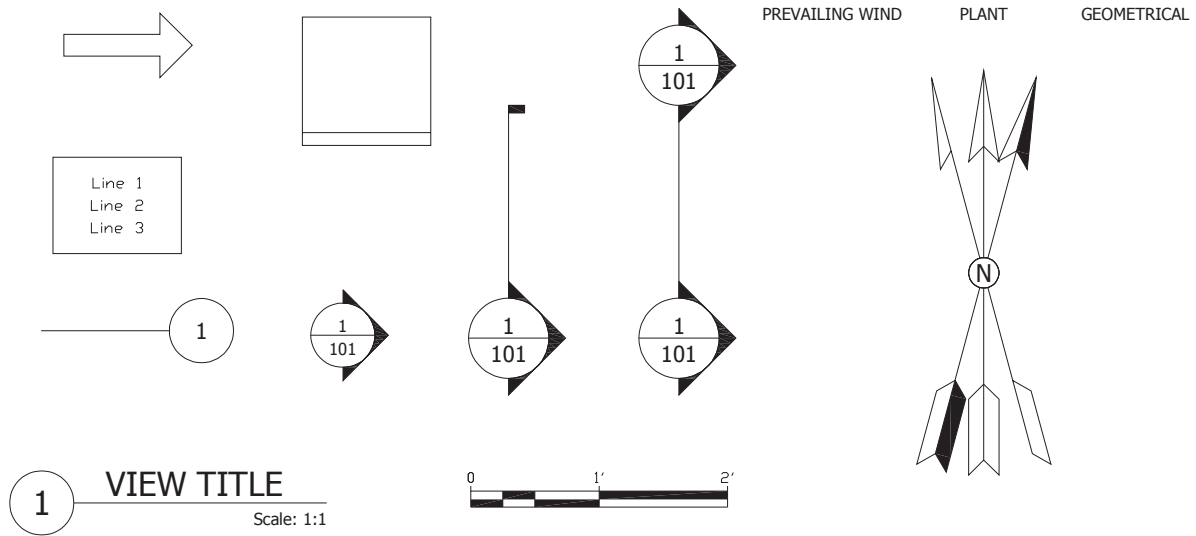
ELECTRONICS: CMOS INTEGRATED CIRCUITS



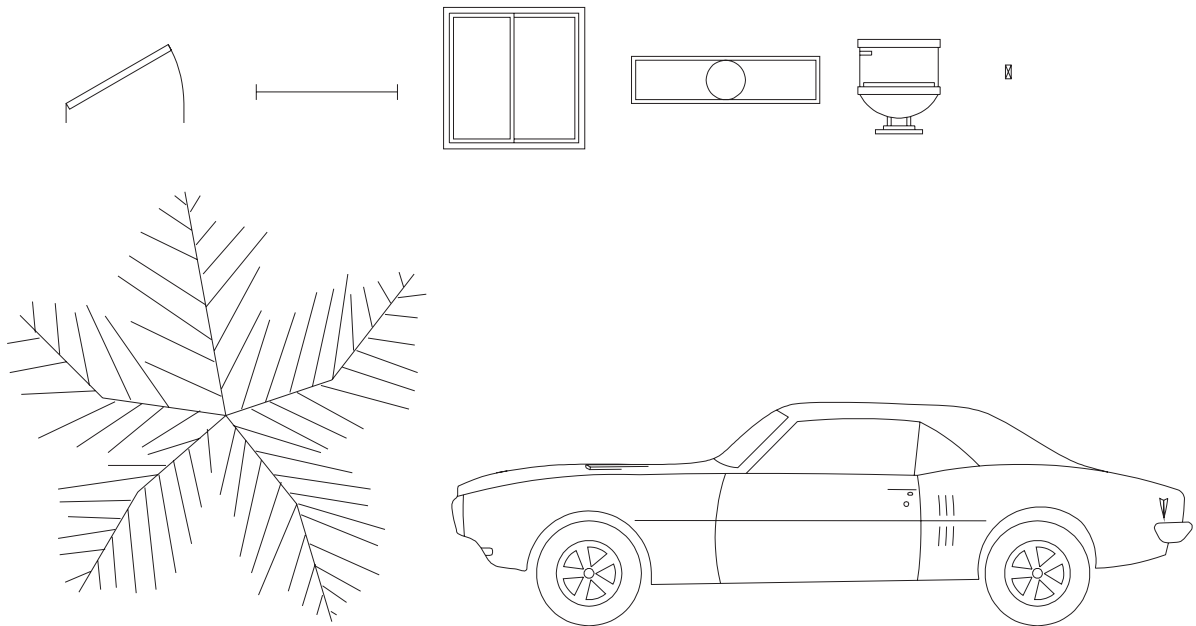
DYNAMIC BLOCK LIBRARIES

The dynamic blocks shown on the following pages are found in the `\autocad 2010\sample\dynamic blocks` folder. A single dynamic block can represent many similar symbols. After being placed in drawings with the `INSERT` command, some blocks can be modified to change their shape; others change their size.

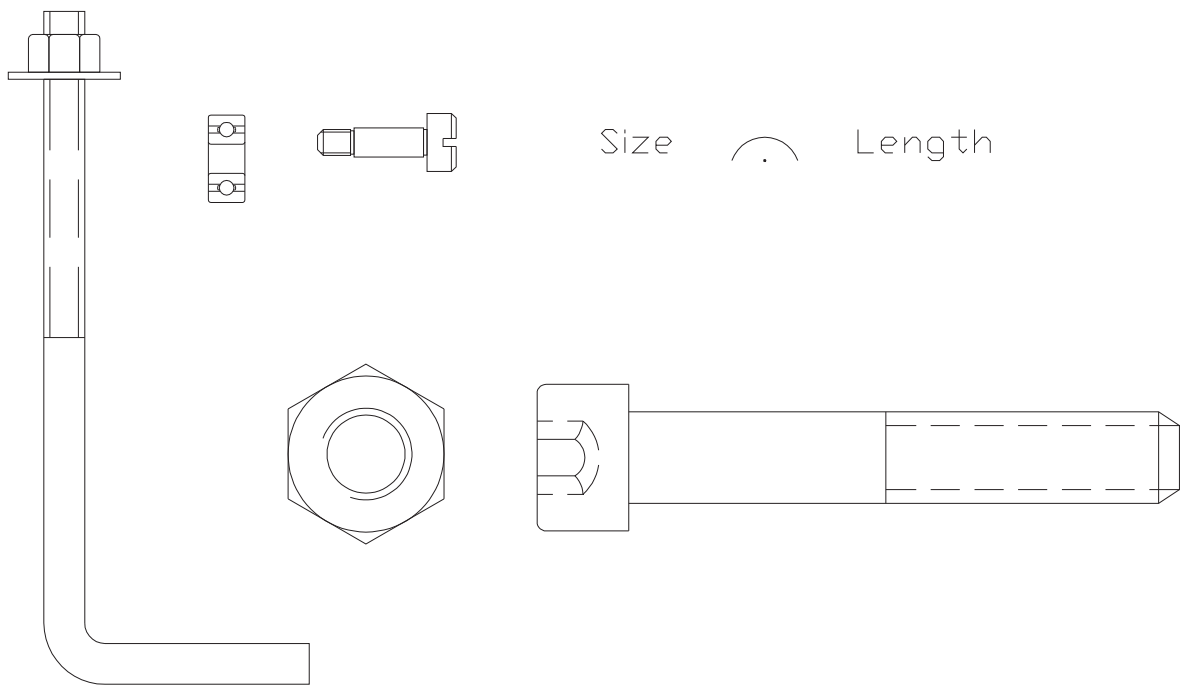
ANNOTATION



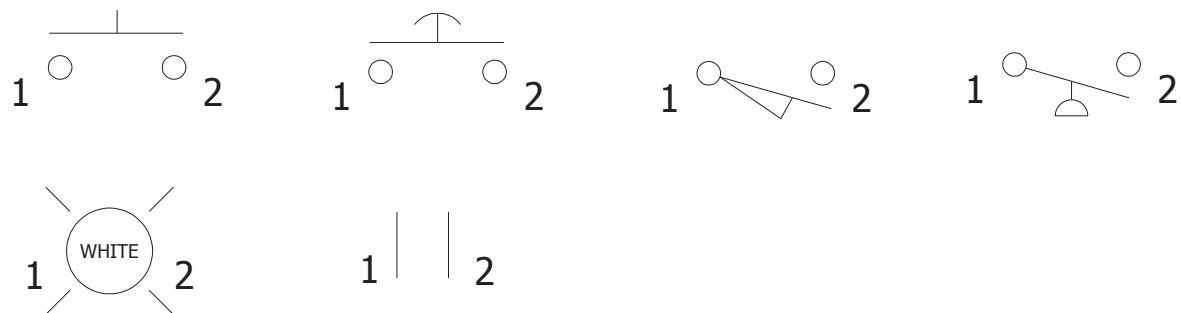
ARCHITECTURAL

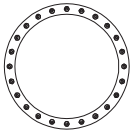
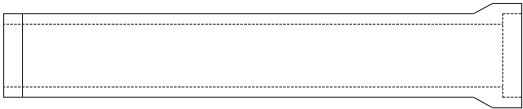
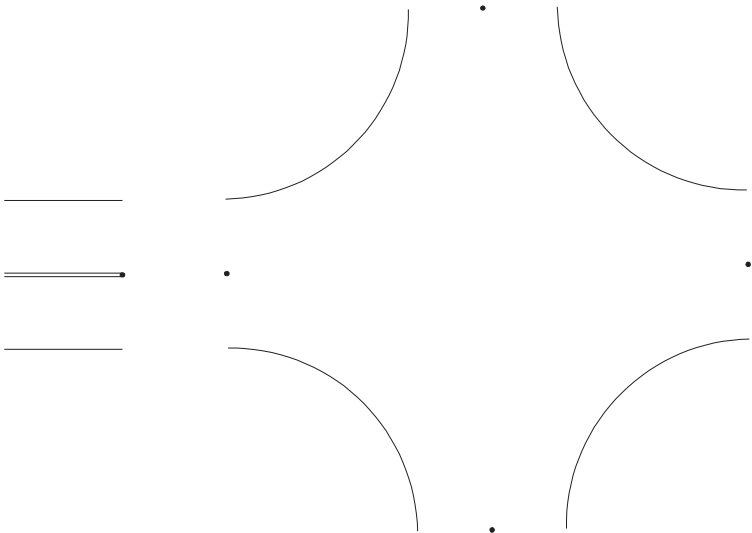


MECHANICAL



ELECTRICAL





I I

AutoCAD Template & Sample Drawings

Template and sample drawings are provided with AutoCAD in drawing *.dwg* or template *.dwt* format.

TEMPLATE DRAWINGS

The following template drawings are found in the hidden *C:\Users\<login name>\AppData\Local\Autodesk\AutoCAD 2010\R18.0\enu\Template* folder. Some templates are DWG, DST, and DGN files (not DWT); most are blank (lack borders).

DWT TEMPLATE FILES

acad -Named Plot Styles.dwt
acad -Named Plot Styles3D.dwt
acad.dwt
acad3D.dwt
acadISO -Named Plot Styles.dwt
acadISO -Named Plot Styles3D.dwt
acadiso.dwt
acadiso3D.dwt
Initial Setup-Architectural-Imperial.dwt
Initial Setup-Architectural-Metric.dwt
Initial Setup-Civil-Imperial.dwt
Initial Setup-Civil-Metric.dwt
Initial Setup-Electrical-Imperial.dwt
Initial Setup-Electrical-Metric.dwt
Initial Setup-Manufacturing-Imperial.dwt
Initial Setup-Manufacturing-Metric.dwt
Initial Setup-MEP-Imperial.dwt
Initial Setup-MEP-Metric.dwt
Initial Setup-Structural-Imperial.dwt
Initial Setup-Structural-Metric.dwt
Tutorial-iArch.dwt
Tutorial-iMfg.dwt

Tutorial-mArch.dwt

Tutorial-mMfg.dwt

DST SHEETSET FILES

Architectural Imperial Sheet Set.dst
Architectural Metric Sheet Set.dst
Architectural Title Block.dwg
Civil Imperial Sheet Set.dst
Civil Metric Sheet Set.dst
Manufacturing Imperial Sheet Set.dst
Manufacturing Metric Sheet Set.dst
New Sheet Set.dst

DWG DRAWING FILES

Generic 24in x 36in Title Block.dwg

DGN DESIGN FILES

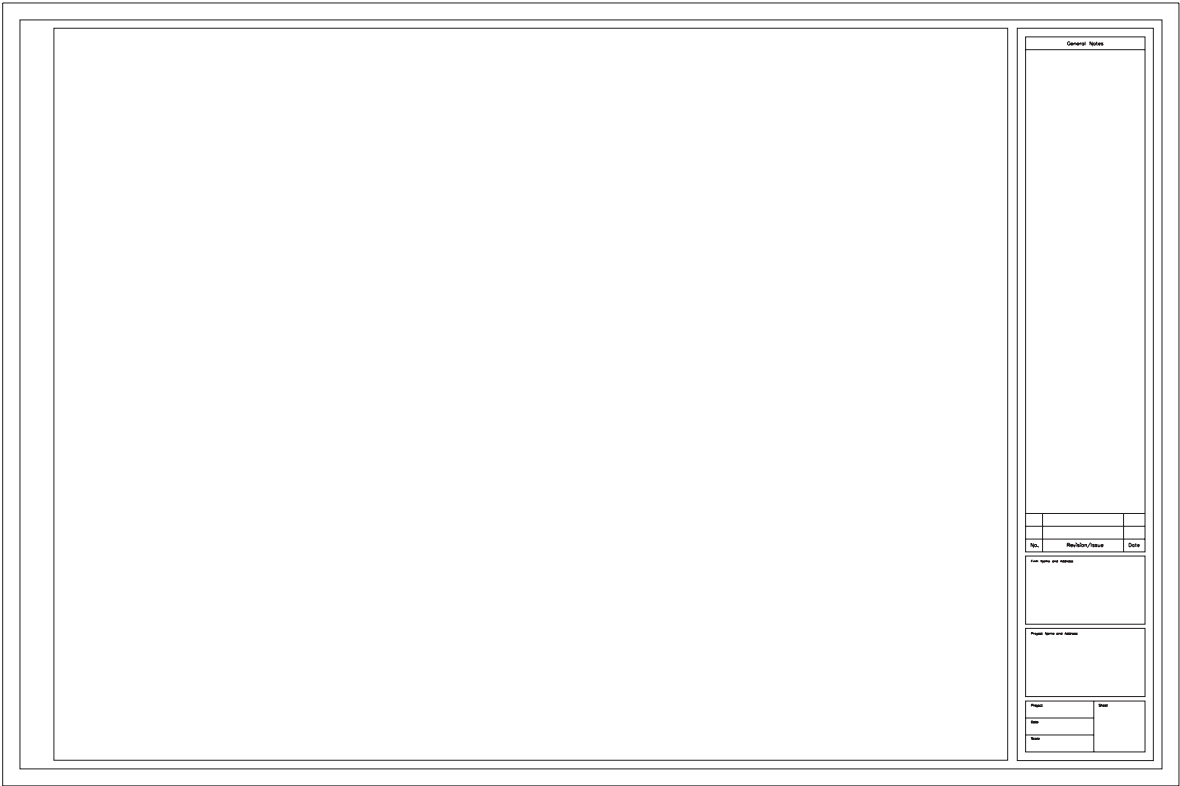
V7-Imperial-Arch-01-Seed2D.dgn
V7-Imperial-Arch-01-Seed3D.dgn
V7-Imperial-Arch-02-Seed2D.dgn

V7-Imperial-Arch-02-Seed3D.dgn
V7-Imperial-Civil-01-Seed2D.dgn
V7-Imperial-Civil-01-Seed3D.dgn
V7-Imperial-Civil-02-Seed2D.dgn
V7-Imperial-Civil-02-Seed3D.dgn
V7-Metric-Seed2D.dgn

V7-Metric-Seed3D.dgn
V8-Imperial-Seed2D.dgn
V8-Imperial-Seed3D.dgn
V8-Metric-Seed2D.dgn
V8-Metric-Seed3D.dgn

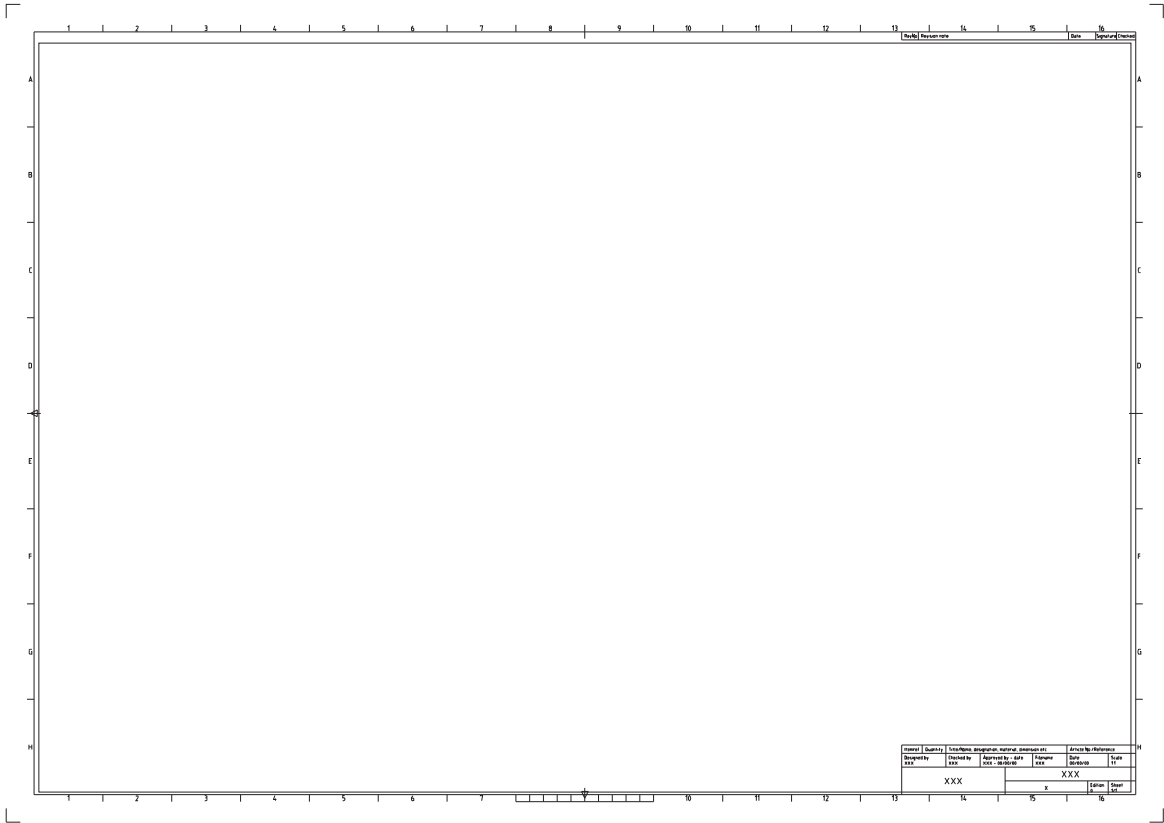
TUTORIAL-IARCH.DWT

Architectural template drawing in imperial units (feet and inches):



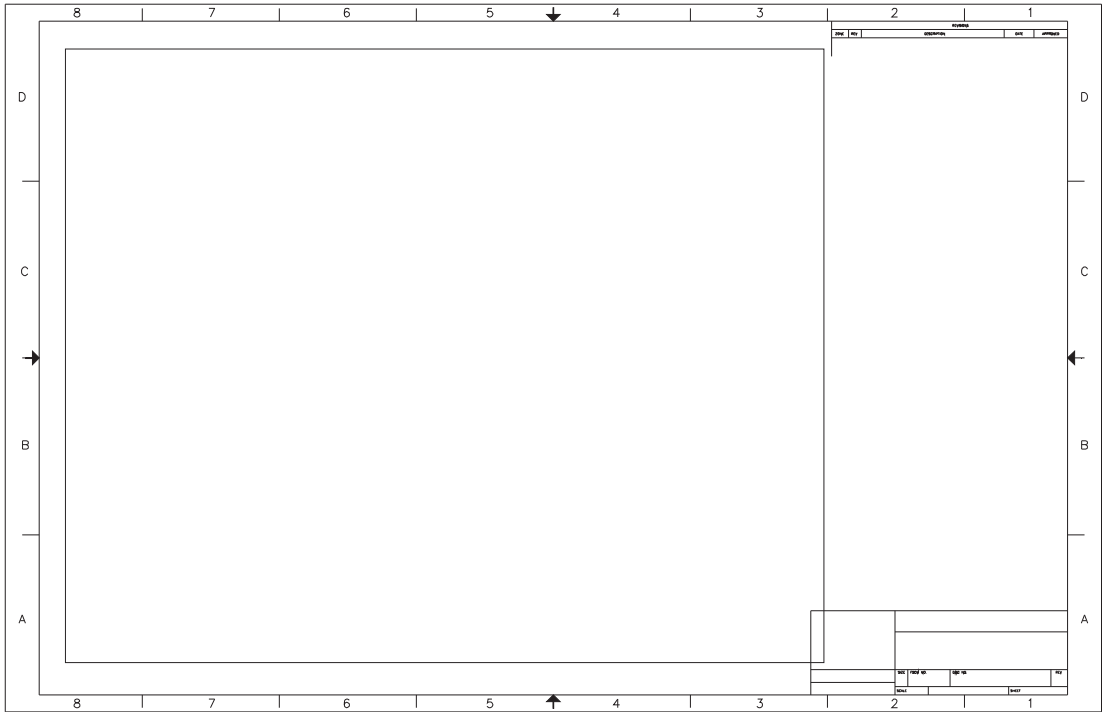
TUTORIAL-MARCH.DWT

Architectural template drawing in metric units (millimeters):



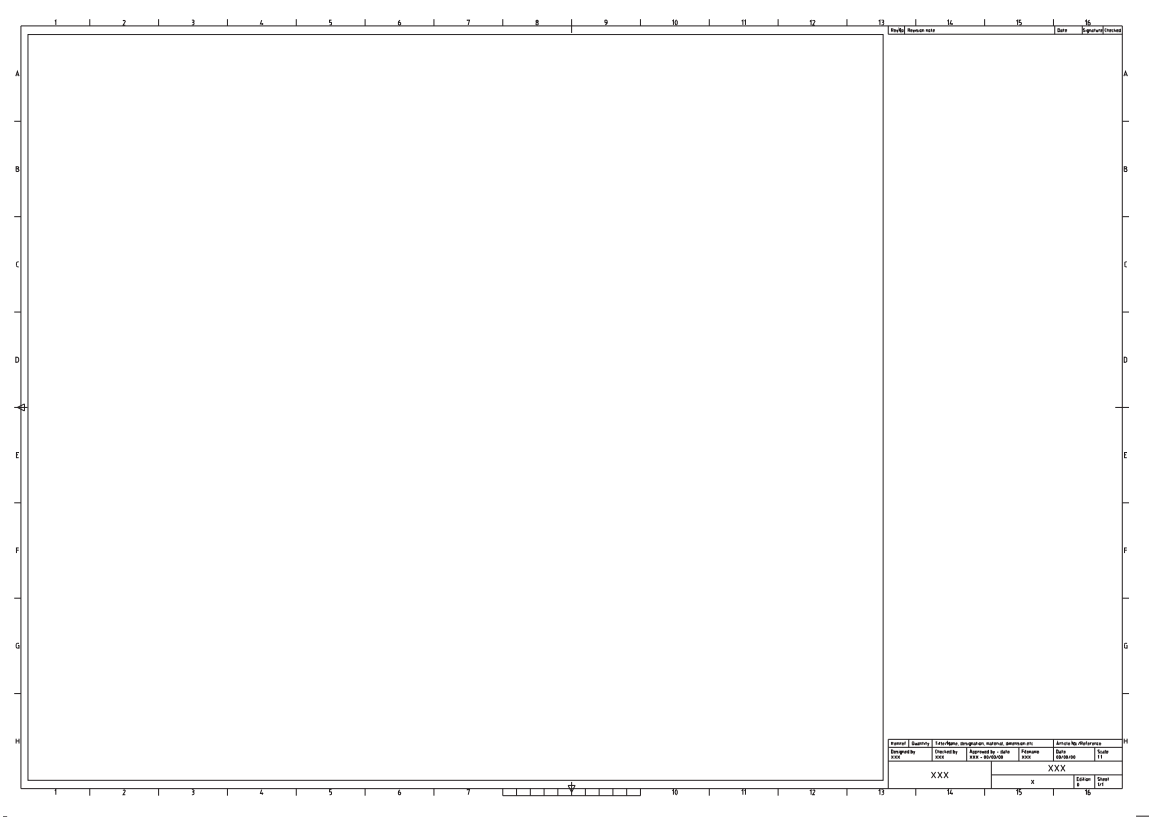
TUTORIAL-IMFG.DWT

Architectural template drawing in metric units (millimeters):



TUTORIAL-MMFG.DWT

Architectural template drawing in metric units (millimeters):

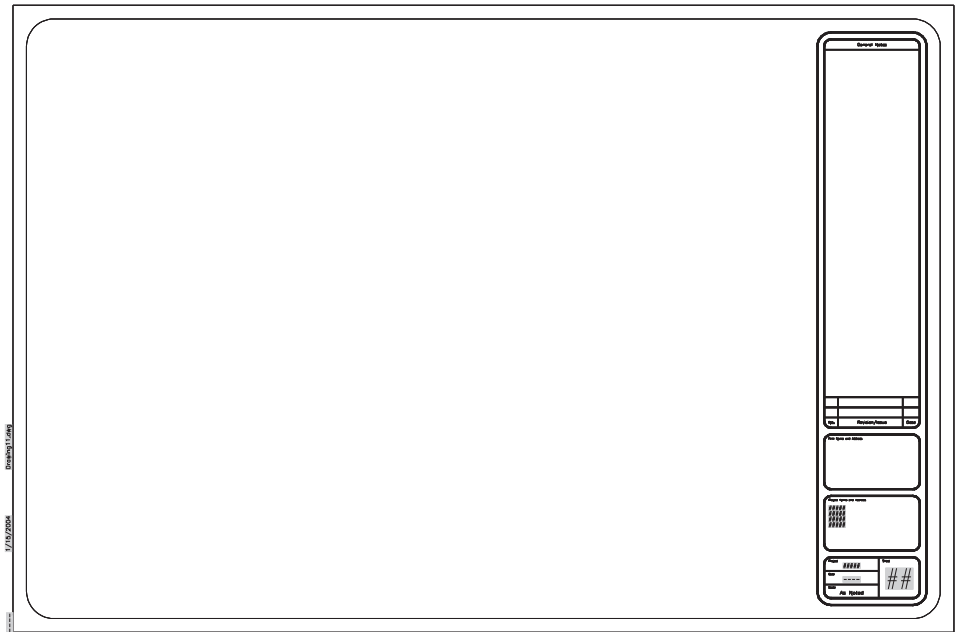


SHEETSET TEMPLATES

The following sheetset template drawings are found in the hidden `C:\Users\<login name>\AppData\Local\Autodesk\AutoCAD 2010\R18.0\enu\Template\SheetSets` folder. Each is available in metric and imperial formats.

ARCHITECTURAL IMPERIAL.DWT

Architectural template sheetset drawing in imperial units. (Metric units template looks similar.)



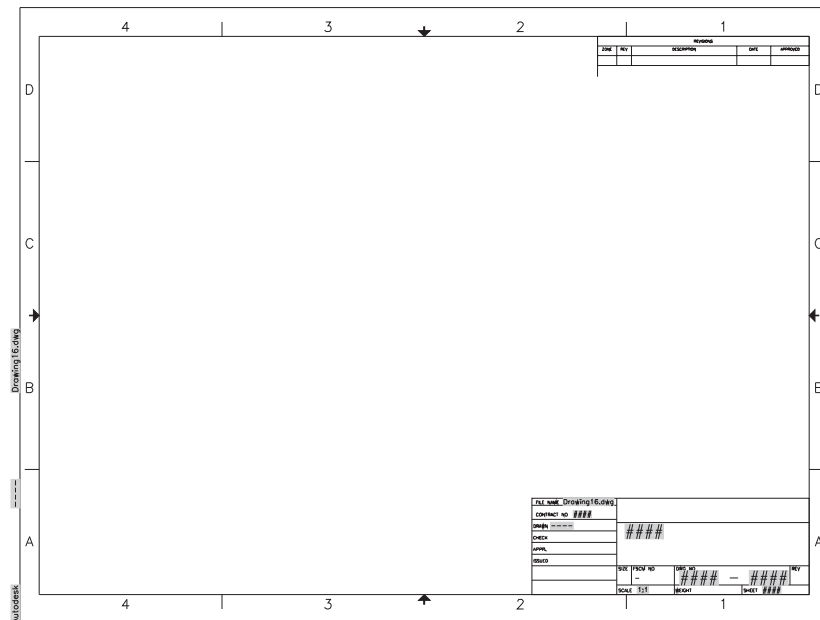
CIVIL IMPERIAL.DWT

Civil template sheetset drawing in imperial units. (Metric units template looks similar.)



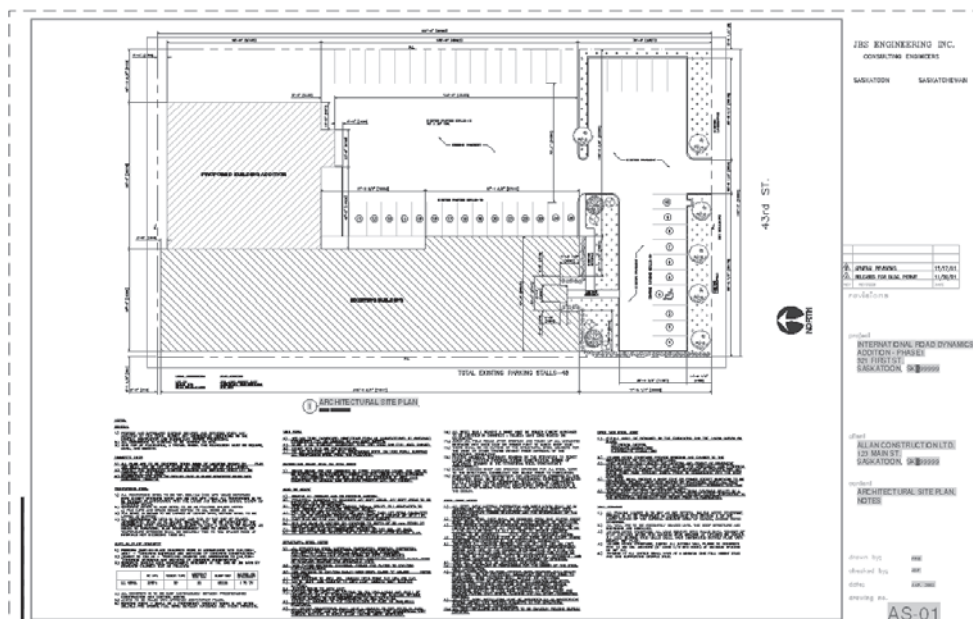
MANUFACTURING IMPERIAL.DWT

Mechanical template sheetset drawing in imperial units. (Metric units template looks similar.)

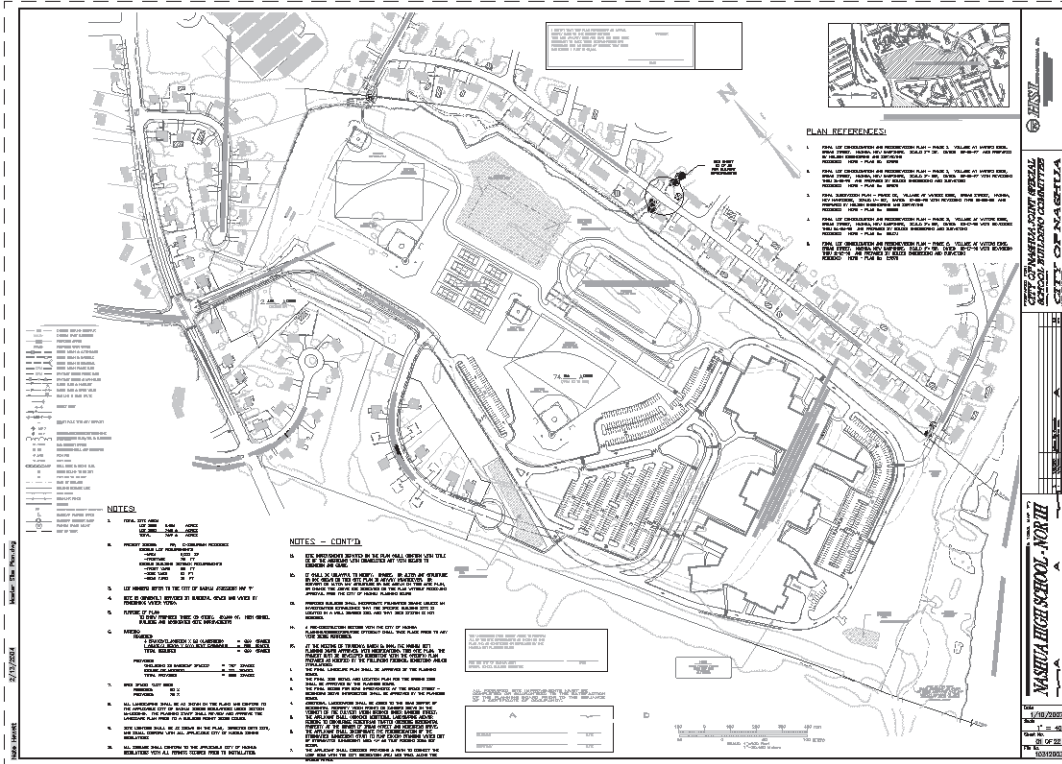


SHEETSET SAMPLES

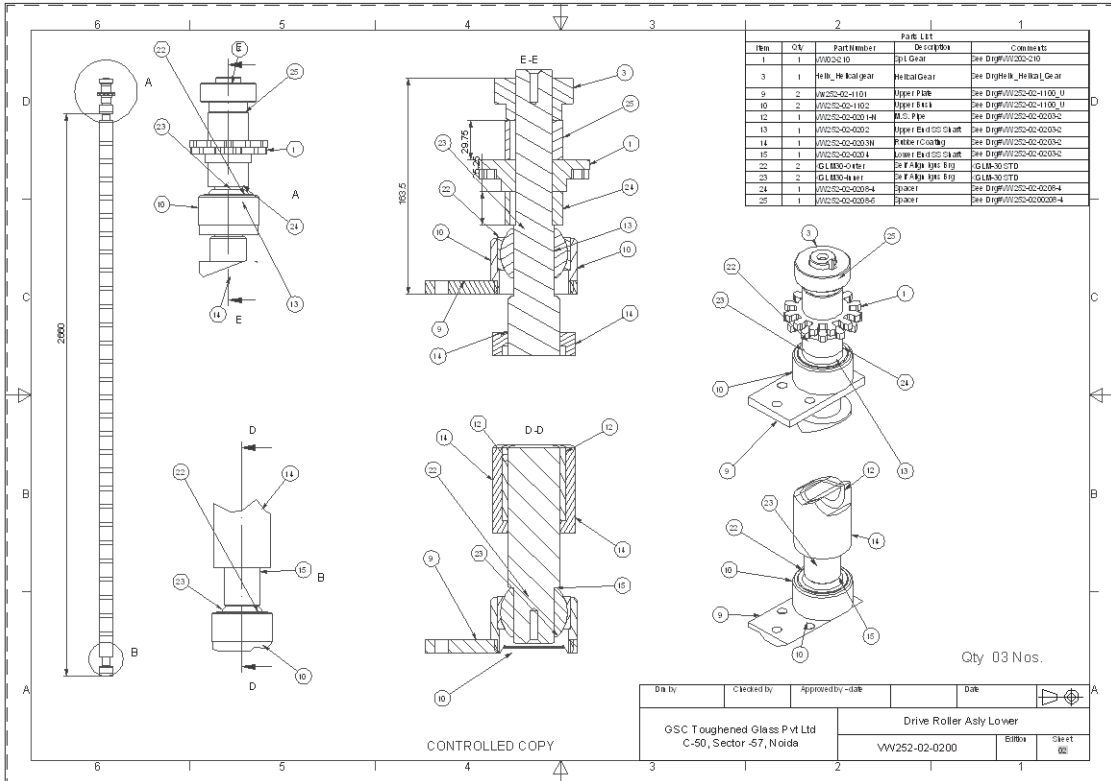
The following are some of the sample sheetsets found in the *AutoCAD 2010\sample\sheet sets* folder.

IRD ADDITION.DST

CIVIL SAMPLE SHEET SET.DST



MANUFACTURING SHEET SET.DST



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Autodesk Resources

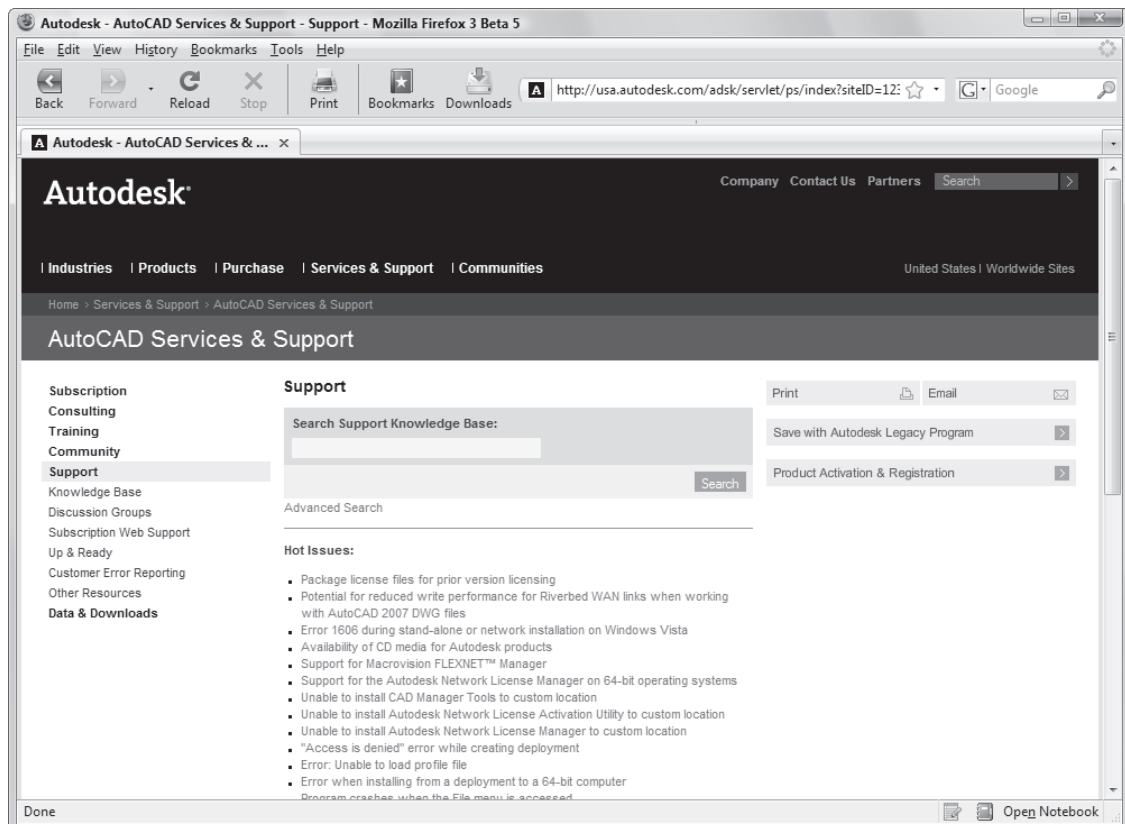
Autodesk provides help files and other forms of assistance with the distribution DVD and on the Internet. Third parties also provide a wealth of information and help.

AUTODESK WEB SITE

The first source of online information is Autodesk's Web page at www.autodesk.com/autocad.

SUPPORT

Click the [Knowledge Base](#) link to access online support in the following categories:



Knowledge Base searches for solutions to problems and bugs (free).

Discussion Groups access moderated discussion groups where users can ask and answer questions among themselves (free).

Subscription Web Support provides additional support for customers who pay an annual fee.

Up & Ready provides 30 days of installation and configuration support if you are in North, Central, and South America, or Japan (free).

Customer Error Reporting describes how AutoCAD's error reporting software works.

Other Resources lists other sources of information and support:

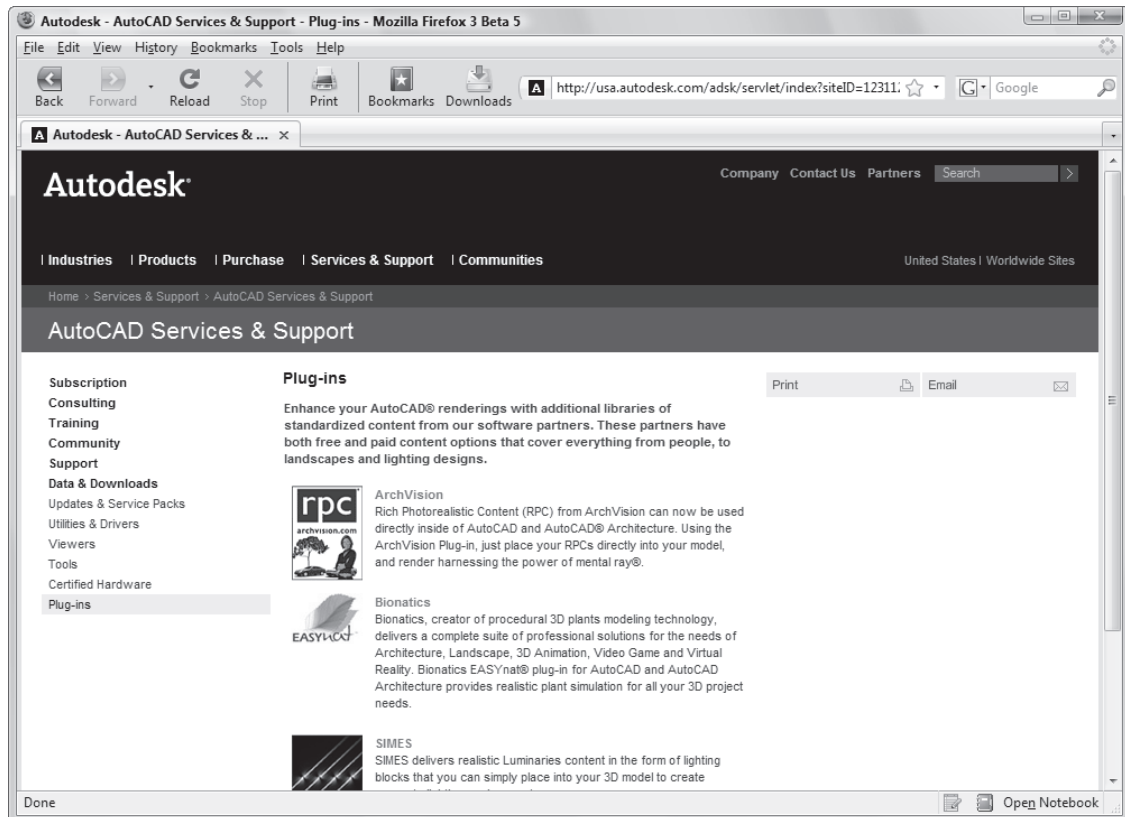
- Autodesk Developer Center (annual fees apply).
- Autodesk Authorized Resellers.
- Autodesk User Group International (free).
- Safety Net.

DATA & DOWNLOADS

Click the [Data & Downloads](#) link to download patches and add-on software. All of these are free:

Updates & Service Packs access service packs and maintenance releases that fix bugs. If you have Communication Center enabled in AutoCAD, then Autodesk notifies you as service packs and hot fixes become available.

Plug-ins are written by third-party developers; some are free, while others require payment.



Utilities & Drivers provide database drivers, peripheral drivers, and Autodesk product utilities.

Viewers list file viewers written by Autodesk:

- DWG TrueView views and plots DWG, DWF, and DXF files; publishes files in DWF format.
- Design Review views, marks up, prints, and round-trips drawings and DWF files.

Tools consist of stand-alone applications:

- Export Layout to Model Space Technology Preview Tool (included with AutoCAD 2010).
- i-drop Indicator modifies Web browsers to initiate the drag-n-drop action.
- Object Enablers access, display, and manipulate object data in applications different from AutoCAD, such as Mechanical Desktop and Civil 3D.
- Units Converter uses conversion factors from the National Institute of Standards and Technology.
- SAMreport-Lite manages network license use of Autodesk software.

Certified Hardware informs you about graphics hardware that takes advantage of all real-time 3D features in AutoCAD.

Autodesk

CompanyContact UsPartnersSearch

IndustriesProductsPurchaseServices & SupportCommunitiesUnited StatesWorldwide Sites

Graphics Hardware List

Graphics Hardware List

The Graphics Hardware List provides you with the status of certified, supported, and nonsupported graphics hardware for the new 3D display features in AutoCAD® 2008, AutoCAD® 2009, and AutoCAD® 2010-based products.

Finding information for your hardware is easy; just choose a card from the list and click the Display List button below. You can also filter the information shown in the list by checking the appropriate box (you can select more than one in each category) and then clicking the Display List button.

Learn more about certified, supported, and nonsupported graphics hardware in the AutoCAD certified hardware FAQ.

Key

Certified cards onlySupported, recommendedSupported, not recommendedNot supported

Choose vendorAll vendors

Choose cardAll cards

Show results for

Certified cards only

All tested cards

Show drivers that are

Supported, recommended

Supported, not recommended

Not supported

Show results for

AutoCAD 2008

AutoCAD 2009

AutoCAD 2010

Show results for

Windows XP

Windows 2000

Windows Vista

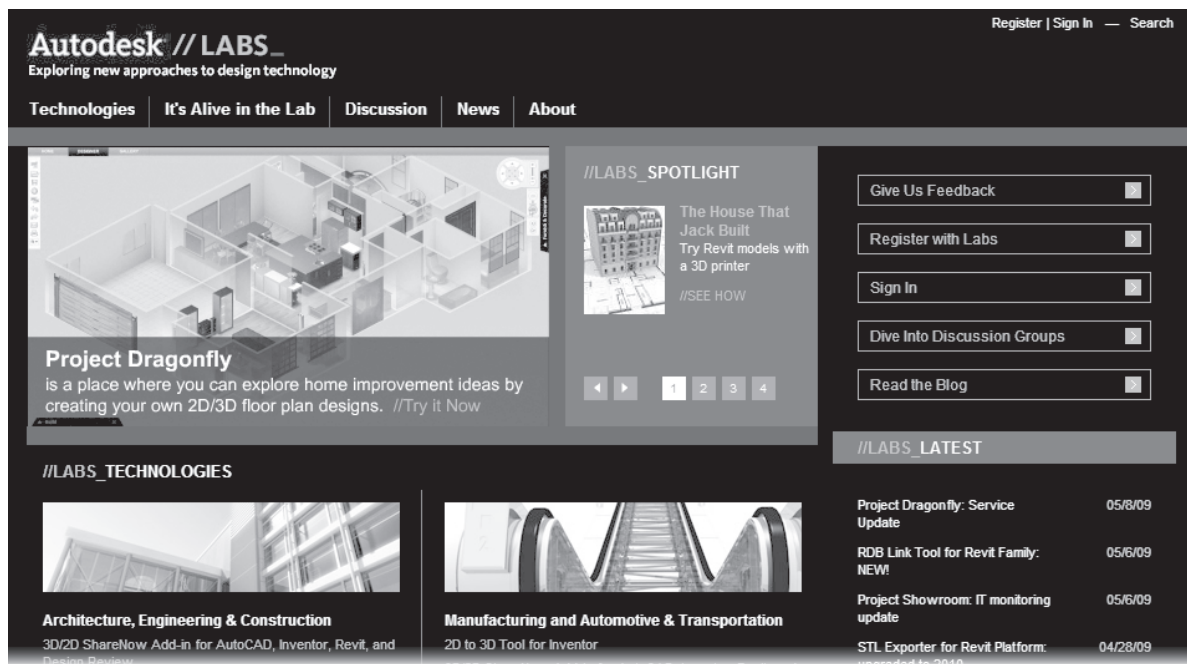
Windows XP 64-bit

Windows Vista 64-bit

Display List

AUTODESK LABS

Autodesk pre-releases unfinished software at its labs.autodesk.com Web site. (Note that the Web address does not use the www.-prefix.)

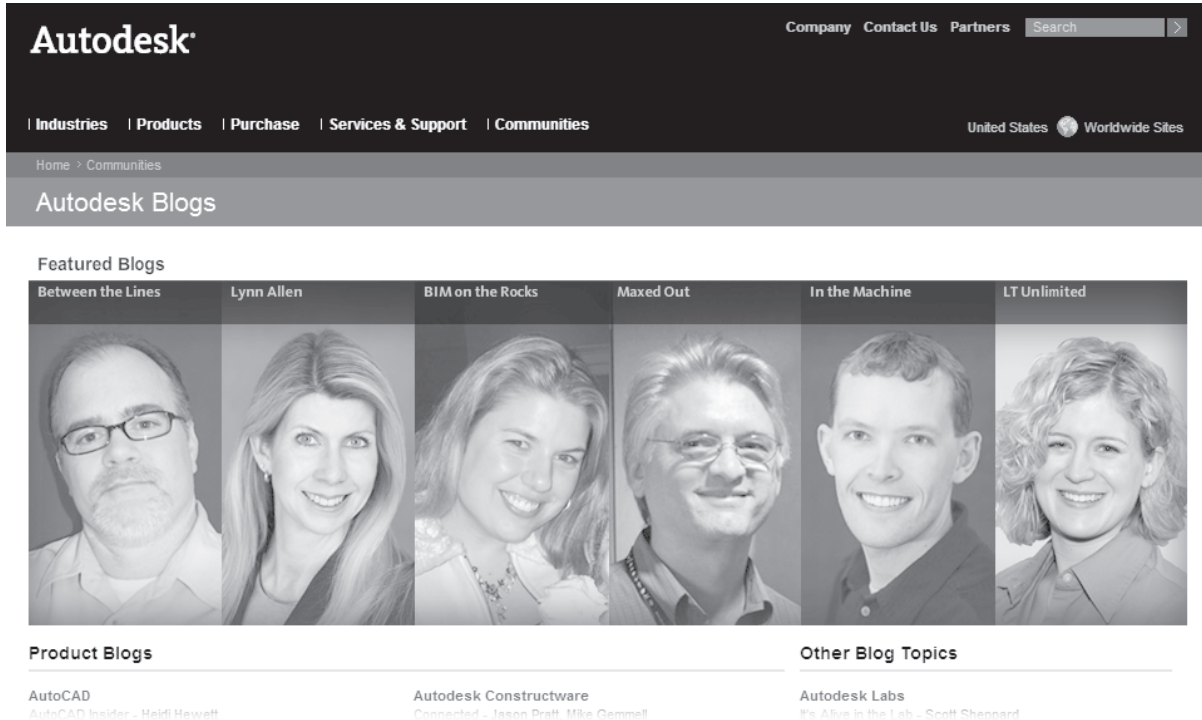


At time of writing this book, the following utilities were available — all free of charge:

- 3D/2D ShareNow Add-in for AutoCAD, Inventor, Revit, and Design Review
- DWG Export for Maya
- Firefox Add-on for Autodesk Design Review
- Google Earth Extension for AutoCAD, Architecture, Civil 3D, Map 3D
- Navigation Tool for AutoCAD
- Project Dragonfly
- Project Draw
- Project Freewheel
- Project Newport
- Project Showroom
- RDB Link Tool for Revit Family
- STL Exporter for Revit Platform 2010
- 2D to 3D Tool for Inventor
- AliasStudio Direct Reader Add-in for Inventor
- CATIA V5 Import Translator for Inventor
- Feature Recognition for Inventor
- Firefox Add-on for Autodesk Design Review
- JT Translator Add-in for Inventor
- Shrinkwrap Add-in for Inventor
- Sustainable Materials Assistant for Inventor

AUTODESK BLOGS

Autodesk has a large number of employees who blog on a regular basis. You find the complete list at its blogs.autodesk.com Web site. By subscribing to a blogs RSS feed, your Web browser alerts you each time the blog is updated. (To do this, click the RSS button found somewhere on the blog page; your browser handles the rest.)



At time of writing this book, these were some of the blogs available:

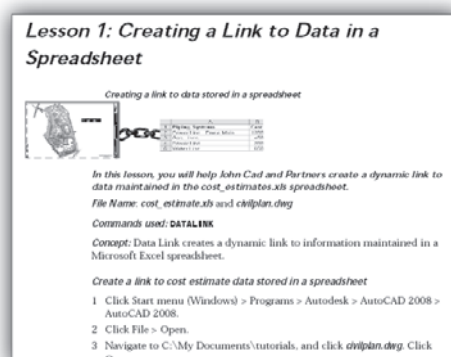
- **AutoCAD Insider** — taking CAD concepts to the next level, hosted by Heidi Hewett
- **Between the Lines** — AutoCAD blog, hosted by Shaan Hurley
- **Between the Poles** — geospatial in the real world, hosted by Geoff Zeiss
- **Between the Walls** — AutoCAD Architecture blog, hosted by Chris Yanchar
- **Beyond the Paper** — DWF blog hosted, by Volker Joseph
- **BIM & BEAM** — Revit Structure blog, hosted by Nicolas Mangon and Wai Chu
- **BLAUG** — Autodesk University blog, hosted by Joseph Wurcher
- **Connected** — blog about design and building project collaboration, hosted by Alex Willingham, Jason Pratt, and Mike Gemmell
- **Controlling the Machine** — AutoCAD Electrical, hosted by Nate Holt
- **The Dave and Dan Civil 3D Show** — Autodesk Civil 3D and the civil engineering marketplace, hosted by Dan Philbrick and Dave Simeone
- **The Dirt** — map it, move it, manage it, roll in it, hosted by Pete Kelsey
- **Drawing the Machine** — AutoCAD Mechanical drafting and design, hosted by Andrew de Leon
- **Duncan's Corner** — tips on creating 3D imagery using Autodesk Maya, hosted by Duncan Brinsmead
- **Engineering Education for the Non-Civil Minded** — engineering education and Autodesk design, hosted by Melissa Thomas
- **From the Ground Up** — a European perspective on design and out of the box thinking with Civil 3D, hosted by Ove Cervin and Jack Strongitharm
- **The Gear Box** - manufacturing solutions, hosted by Jay Tedeschi

- **Geospatial Made in France** - hosted by Gwenaël Bachelot
- **In the Machine** - Inventor, hosted by Garin Gardiner
- **In the Pipes** - Autodesk Plant Solutions blog, hosted by Ursula Sadiq
- **Inside Hues** — behind the scenes of Autodesk Impression, hosted by Lisa Crounse
- **Inside the System** — covering the mechanical, electrical, and plumbing disciplines and how they relate to building information modeling and sustainability, hosted by Kyle Bernhardt
- **It's Alive in the Lab** — Autodesk Labs blog, hosted by Scott Sheppard
- **Lynn Allen's Blog** — hosted by Lynn Allen
- **Maxed Out** — 3ds Max, hosted by Kelcey Simpson and Ken Pimentel
- **Mayalicious** — Maya, hosted by Cory Mogk
- **MotionBuilder** — hosted by Curtis Garton
- **Off Course, On Target** — where unexpected paths lead to great discoveries, hosted Wayne Hodgins
- **Outside the Lines** — Impression blog, hosted by Todd Hunter
- **Plan-It Autodesk** — blog for and about project planners, hosted by Gene Roe
- **Points of Interest** — blog for location-based service developers, hosted by Location Services Developer Team
- **Through the Interface** — developer blog, hosted by Kean Walmsley
- **The Topobase Insiders** — Topobase blog, hosted by Cedric Moullet and Andreas Boos
- **Under the Hood** — Vault and Productstream, hosted by Brian Roepke
- **Wicked Cool Stuff** — blog on all the Wicked Cool Stuff in the Civil 3D world, hosted by Nick Zeeben
- **2D to (Civil) 3D** — civil 3D for Education Community, hosted by Sanjay Asnani
- **3D Roadie** — stream of consciousness about 3D, hosted by Shawn Hendriks

AUTODESK TUTORIALS

Autodesk provides a collection of tutorials at its support.autodesk.com site: from the **Browse Training by Product** droplist, choose “AutoCAD,” and then click on [Tutorials](#), and then on [Feature Tutorials](#). At the time of writing, these were some of the dozens of tutorials available at no charge — typically in the form of PDF documents and sample drawings:

- Customize the Ribbon
- Working with Action Recorder
- Visualization: Basics of Rendering



- Create Basic 3D Objects
- Create 3D Objects from 2D Objects

- Create a 3D Environment to Draw 3D Models
- Control the Workplane

AUTODESK USER GROUP INTERNATIONAL

Autodesk supports an international user group, which puts on the annual Autodesk University event each year, collects wishlist items, and otherwise supports CAD users. Membership in the AutoCAD User Group International is free at www.augi.com.

AUGI Autodesk User Group International

Product Communities Forums Publications Education Chapters Surveys Volunteering Autodesk University

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Want to make a difference?
Now is the time to submit your nomination to be an AUGI Director
[Click here for more info...](#)

AUGI 2009 Board of Directors - Nominations are Open

Now accepting nominations for the 2009/2010 AUGI Board of Directors! Do you want to make a difference in the future of AUGI? Click here to learn more about the nominations process and how to submit your nomination. Nominate yourself or others. It is easy! Hurry, nominations close at midnight on May 24th, 2009.

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Password:

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 >> [Click here to join!](#)

Featured Links

Autodesk User Group International Blog
 Blaugi - Official Blog of Autodesk User Group International

AUGI Survey

AUGI's monthly survey polls AUGI members for their opinions on a number of topics, from Autodesk product specifics, to AUGI programs, and lots more. The results of the monthly surveys enable AUGI to engineer programs that meet members'

My AUGI

Visit My AUGI from time to time and update your member profile, including your email address, industry affiliation, and preferences. Pick the AUGI publications you wish to receive and the local chapters you wish to join. You can

Announcements

The April issue of AUGI HotNews is available online. Read it today!

Don't miss the January/February issue of AUGIWorld. Get the *Green* edition today!

Quick Links

AUGI Training Program (ATP) Courses
 My ATP
 Current AUGI Survey
 AutoCAD Wish List
 Civil 3D Wish List

At time of writing this book, the Top Ten wishlist for AutoCAD included the following:

1. Design File Locking
2. Block Manager
3. Option to Highlight Overridden Dimensions
4. Selection Set to Remain While Zooming and Panning
5. Isometric Tools
6. Convert Mtext into a Multileader
7. Specific Levels of Draw Order
8. Option for Spell Check in Right-click Menu
9. Associate a Layerstate to a Viewport
10. Quickly View What's on a Layer

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